Alibre Design User Guide

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CHAPTER 1

Installation

Thank you for choosing Alibre Design! We welcome you to the ever-growing community of designers who have discovered the high value of Alibre Design's feature-rich modeling capability.

Alibre Design is easily installed on your computer. Use these directions to get up and running in a matter of minutes.

In This Chapter

System Requirements .....
On the DVD .....
Installing Alibre Design .....
Uninstalling Alibre Design .....

1.1 System Requirements

The following requirements must be met to install and run Alibre Design.

Supported Operating Systems

- Windows® XP Professional or Home Edition
- Windows Vista Business, Home, or Ultimate

Unsupported Operating Systems

- Windows XP Starter Edition
- Windows NT
- Windows Me
- Windows 98

Other Operating Systems

- Windows XP x64 / Windows Vista x64*
- Windows 2000, Server 2003**

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* Occasionally, some display-related issues may occur, depending on the system setup. However, Alibre Design does work on these operating systems. Please contact support if you have having issues.

** Alibre Design may function on these operating systems; however, it has not been tested. It is strongly recommended to move to one of the supported operating systems.

Software Requirements

- Internet Explorer 6.0 or later
- .NET 2.0 Framework - part of the installation process
- J# Redistributable - part of the installation process

> **Note:** Alibre Design is only compatible with J# 2.0. If J# Second Edition is installed, it will need to be uninstalled before Alibre Design can be used. During the installation process of Alibre Design 11.0, J# 2.0 will be installed automatically.

- Microsoft DirectX 9.0c

Minimum Hardware Requirements

- Intel Pentium or equivalent processor; 2 GHz or faster
- RAM: 1-2 GB
- 1024 x 786 minimum screen resolution; 1280 x 1024 or greater recommended
- 64 MB or higher video card with DirectX 9.0c
- Color: 32-bit
- Hard Disk Space: 500 MB
- Virtual Memory: 1 GB
- CD-ROM drive
- Mouse or pointing device

Additional Recommendations for assemblies of 500 or more parts

- Intel Pentium or equivalent processor: 3 GHz or faster
- RAM: 2 GB or more
- Virtual Memory: 2 GB

Recommended changes for Windows Vista

- Intel Pentium or equivalent processor: 3 GHz or faster
- RAM: 2 GB or more
- Video Card: 128 MB RAM or higher
- Color: 32-bit

1.2 On the DVD

The following items are on the installation DVD:

- Alibre Design 11.0
- Alibre Design Help
- Alibre PhotoRender
- DirectX 9.0c
- Internet Explorer 6.0
- ALGOR DesignCheck
- Alibre Part Library
- Alibre CAM
- Alibre Motion
- Machinist Toolbox
- Engineers Toolbox
- Alibre Vault by M-Files
- TraceParts Access Instructions

1.3 Installing Alibre Design

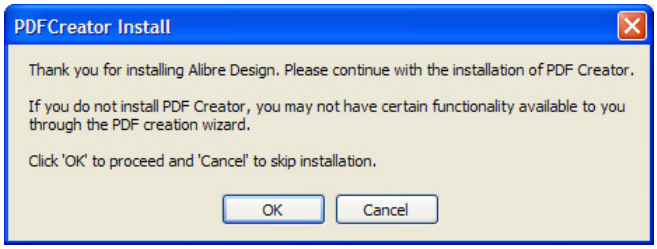
The Alibre Design Installer is available on the installation DVD as well as from the Alibre website.

➢ ***To download the Alibre Design installer from the website:***

1. Sign in with your user name and password and go to the Downloads page.
2. Click Alibre Design Client to download the version of Alibre Design associated with your license. We recommend that you save the installer to your hard drive in case the installation process is interrupted.
3. Once you have saved the installer to your hard drive, double click the file to begin the installation process.
4. Continue the installation as described below, beginning with step 4.

➢ ***To Install Alibre Design from the DVD:***

1. Insert the DVD. Within a few seconds, a window appears. If the window does not appear, open the Windows Explorer, and double-click your DVD drive icon to open the window.
2. Click **Customers** on the DVD menu.
3. Select your product level.
4. Click **Install Alibre Design 11.0**. The Alibre Design installation begins.
5. Follow the prompts in the Installation Wizard.
6. By default, Alibre Design is installed in the **Program Files** directory on the **C:** drive. If desired, select a different destination folder.
7. Choose where to place shortcuts.
8. Select the language you prefer.
9. Click **Install**.
10. When the installation is complete, a message will appear prompting you to install PDFCreator. Click **OK** to continue.


PDFCreator Install
Thank you for installing Alibre Design. Please continue with the installation of PDF Creator.
If you do not install PDF Creator, you may not have certain functionality available to you through the PDF creation wizard.
Click 'OK' to proceed and 'Cancel' to skip installation.
OK
Cancel

Note: PDFCreator cannot be installed on the Vista operating system. In addition, if you do not install PDFCreator, you will be unable to fully publish BOMs or Drawings to PDF. If you choose to install a different PDF Printer application, you can print to PDF, but you will be unable to add a header or footer to the PDF.

11. Select the language you want to use and click **OK**.
12. Click **Next** in the Welcome Dialog.
13. Read the license agreement, then check **I accept the agreement** and click **Next**.
14. Choose the Standard or Server Installation, then click **Next**.
15. Enter a name for the **PDF Printer** (the default is PDFCreator), then click **Next**.

16. Select the **Location** to install PDFCreator, then click **Next**.
17. Read the information in the PDFCreator Toolbar dialog, then click **Next**.
18. Choose the **components** you want to install, then click **Next**.
19. Choose where to place the program's shortcuts, then click **Next**. (If you do not want a shortcut, check the **Don't create a Start Menu** folder option.)
20. In the Select Additional Tasks dialog, check the items that you want performed during the installation, and then click **Next**.
21. Click **Install** in the Ready to Install dialog.
22. Click **Finish** to exit the wizard. The Alibre Design Install Wizard will indicate that the installation is complete.
23. Click **Finish**.

1.4 Uninstalling Alibre Design

➢ ***To Uninstall Alibre Design:***

1. From the **Start** menu, select **Control Panel** or **Settings - Control Panel** depending on your operating system.
2. Select **Add or Remove Programs**.
3. From the **Currently installed programs** list, select **Alibre Design**.
4. Click **Remove**.
5. At the prompt, click **Yes** to confirm that Alibre Design should be removed.

   The uninstall program removes program files, folders, and registry entries.
6. When the files are removed, the uninstall program may indicate that the process is complete. Click **OK**.

CHAPTER 2

Getting Started With Alibre Design

Alibre Design is a powerful mechanical design software application. You can use Alibre Design to create complex 3D designs and 2D drawings. This chapter provides a high-level overview of the basic design capabilities in Alibre Design.

In This Chapter

Initial Launch of Alibre Design .....
Tutorials .....
The Home Window .....
Workspaces .....
Common Terms .....
Toolbars .....

2.1 Initial Launch of Alibre Design

➢ ***To launch Alibre Design:***

From the Start menu, select **All Programs** > **Alibre Design**. An initial startup screen is displayed.

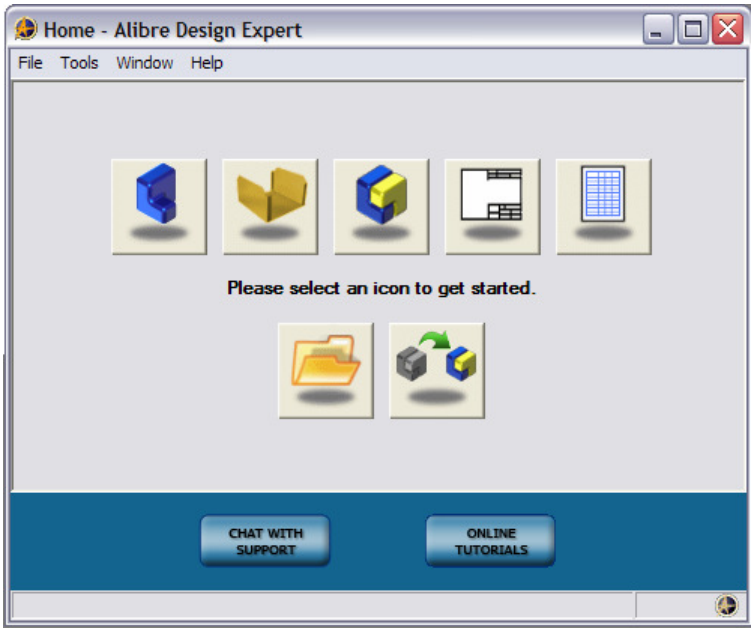

Home - Alibre Design Expert
File Tools Window Help
Please select an icon to get started.
CHAT WITH SUPPORT
ONLINE TUTORIALS

Figure 1: Startup Screen

If you have already registered with Alibre, Inc.…

1. Enter the **user name** and **password** you obtained from Alibre, Inc.

> **Note:** Shortly after purchasing Alibre Design, new customers receive an email from Alibre containing a user name and a temporary password. If you register and/or purchase on the website, your user name and password are displayed at that time.

2. Click **Submit**. If an internet connection is detected, your license key will be automatically retrieved from the Alibre server. Otherwise, you will see a message with a 5 character Site Key and instructions to follow to obtain a license key.

   Note: If you call Alibre Support for help in obtaining your license key, you will have to provide this 5 character Site Key, as well as your computer's Machine Name.

3. After several seconds, the startup screen will be replaced by the Alibre Design **Home Window**. Also, the **Set Default Workspace** dialog will appear.

4. Choose which default workspace you would like to see when starting Alibre Design from the drop-down menu. You can select **None** if you do not want any default workspace to open.

If you have not yet registered with Alibre, Inc.…

Choose one of the available options:

- **Register Alibre Design Xpress** - Register your free copy of Alibre Design Xpress to gain additional functions. You will be directed to a web form where you can register with Alibre, Inc. You will select a user name and be provided with a password to enter in the startup screen shown above.
- **Buy Alibre Design Products** - Purchase Alibre Design software, training materials, and other optional components.

2.2 Tutorials

Alibre Design tutorials are integrated with the application. To access the tutorials, press **F1** to open the Help system; then, select the **Tutorials** book.

We strongly recommend that you work through these included tutorials. Working through them will give you a good understanding of the basic modeling terms and concepts used in Alibre Design. You will also become familiar with 3D viewing techniques in Alibre Design.

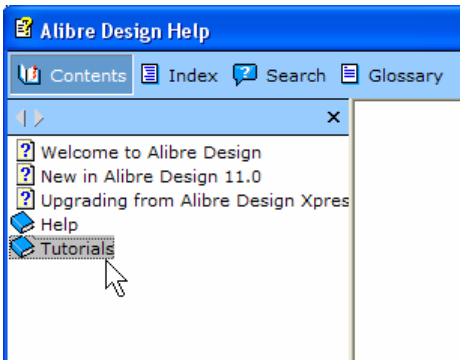


Figure 2: Tutorials

2.3 The Home Window

Alibre Design is comprised of four main components: the **Home window**, **Design Workspaces**, **Drawing Workspaces**, and **BOM Workspaces**. Each component opens in a separate window that you can independently resize, position or tile.

When you launch Alibre Design, the Home window appears first and is always open while Alibre Design is running. The Home window serves as the starting point for all other areas of Alibre Design.

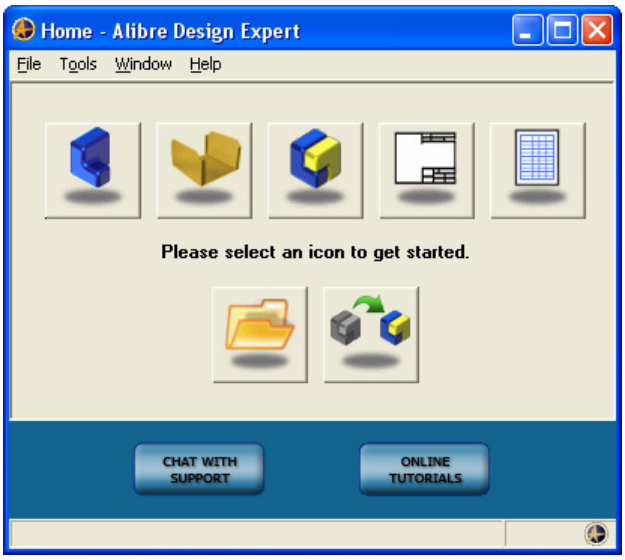

Home - Alibre Design Expert
File Tools Window Help
Please select an icon to get started.
CHAT WITH SUPPORT
ONLINE TUTORIALS

Figure 3: Alibre Design Home Window

2.3.1 System Options

A number of system options can be set and modified from the Home window. From the **Tools** menu select **Options**. The **Options** dialog appears.

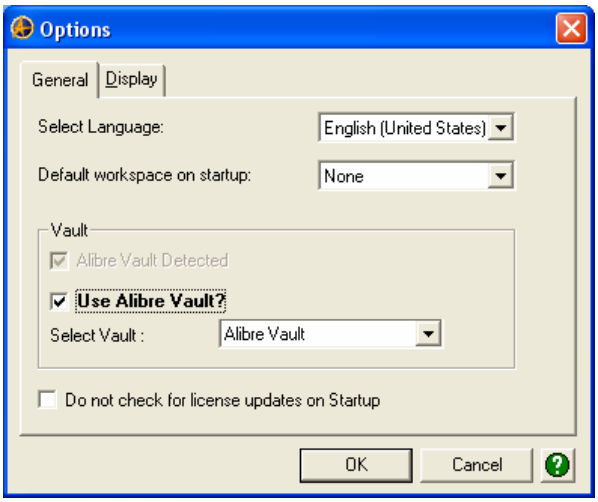


Figure 4: System Options Dialog

General tab:

The General tab allows you to specify the language you are using, which type of workspace opens by default on startup (you can select **None** so that no workspaces automatically open upon launching the software), and your Alibre Vault settings (if the Vault is installed and you are licensed for it).

Also, you can change the startup options for checking for software updates and you can choose to have Alibre Design remember your password.

Display tab:

Under the **Display** tab you can check your graphics settings by selecting the **Settings** button.

2.4 Workspaces

Before proceeding further in this User Guide, we strongly recommend that you work through two of the basic Part Design tutorials, *Introduction to Sketching and Constraints,* **and** *Modeling a Simple Part,* **found in the Help system.**

Working through these tutorials will give you a good understanding of the basic modeling terms and concepts used in Alibre Design. You will also become familiar with 3D viewing techniques in Alibre Design.

By completing these tutorials now you will be better prepared for the material in the subsequent sections.

All design related tasks are carried out in windows referred to as workspaces. New workspaces can be opened from the Home window or other open workspaces. From the **File** menu, select **New** to open a new workspace. Five workspace types are available: part, sheet metal part, assembly, drawing, and bill of materials.

Note: Sheet metal part, drawing, and bill of materials workspaces are not available in all versions of Alibre Design.

The **Design Explorer**, displayed on the left side of a part or assembly workspace, lists all reference geometry, such as planes and axes, and feature geometry associated with the design. Toolbars are located on top and to the right of the main work area.

The workspace can be customized a number of ways. From the **Tools** menu, select **Options**. Use the **Color Scheme** tab to change the background color of the main work area. Default colors can be selected and custom colors can be added. Additionally, the toolbars can be positioned and toggled on and off - from the **View** menu, select **Toolbars**.

To set workspace properties, from the **File** menu, select **Properties**. The Design Properties dialog includes six tabs.

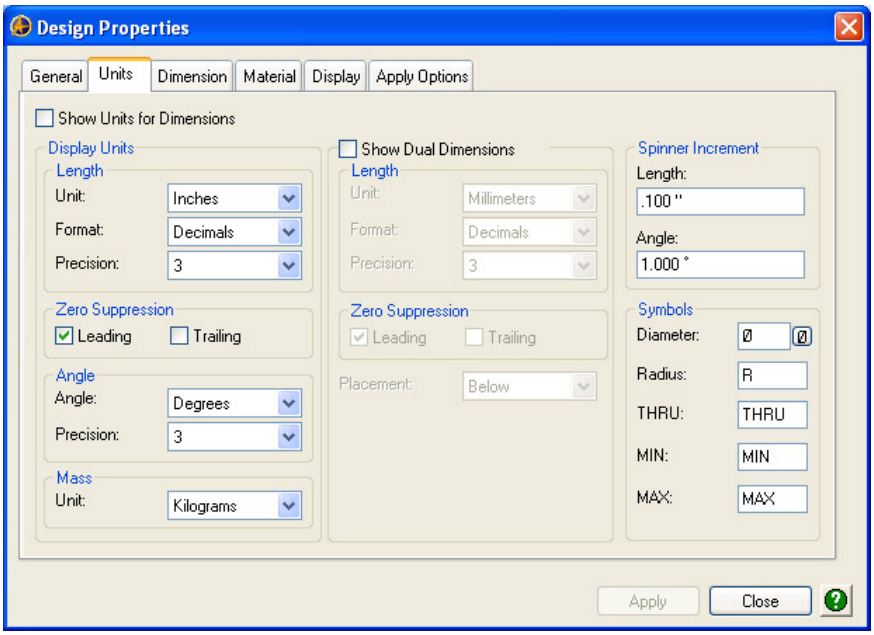


Figure 5: Design Properties Dialog

The **General Tab** is used to fill in product or design data. This data can be used to automatically populate title blocks or BOMs.

The **Units** tab is employed to set all design units and associated display format and precision. Several other unit display settings are also available.

The **Dimension** tab is used to control dimension properties such as size, dimension orientation and spacing, and arrowhead style.

The **Material** tab is used to set the density of the part material. This information is used when calculating physical properties of the part.

The **Display** tab is used to control the graphics display.

- In the **Display Acceleration** field, you can set the options you want to use when in Display Acceleration mode in parts and assemblies. Please see the *Display Acceleration* (on page 183) section for more detail.
- The **Curve Smoothness** setting is applied on this tab to set the precision of the graphic display. This does not affect the precision of the model - only the display of the model on your screen. Lower values will make circular items appear coarse, but performance will improve. Large values improve the visual appearance, but can slow down performance. Reasonable settings for this are usually between 30 and 50.

The **Apply Options** tab is used to apply Design Properties settings on a system-wide basis, or only to the current workspace.

2.5 Common Terms

Here are some of the most common terms you will need to be familiar with when starting out with Alibre Design:

Home Window: The Home Window is the starting point for all activities in Alibre Design. This is the first window to open, and is always open while Alibre Design is running.

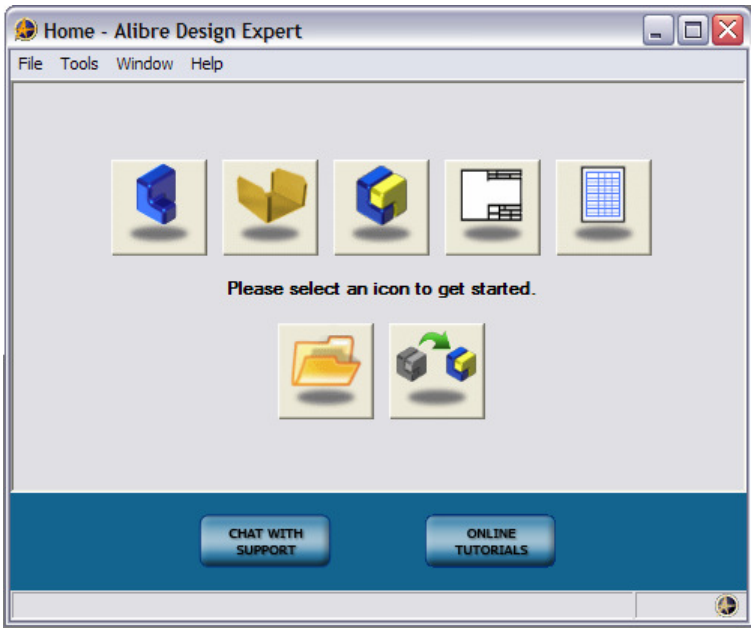


Figure 6: Startup Screen

Workspace: Windows where design related tasks take place. You can have multiple workspaces open at one time. Each part, sheet metal part, assembly, drawing, or bill of material has a separate workspace.

Work area: The area in a workspace where sketches and features are created.

Design Explorer: The area in a workspace where the history of the model creation is listed.

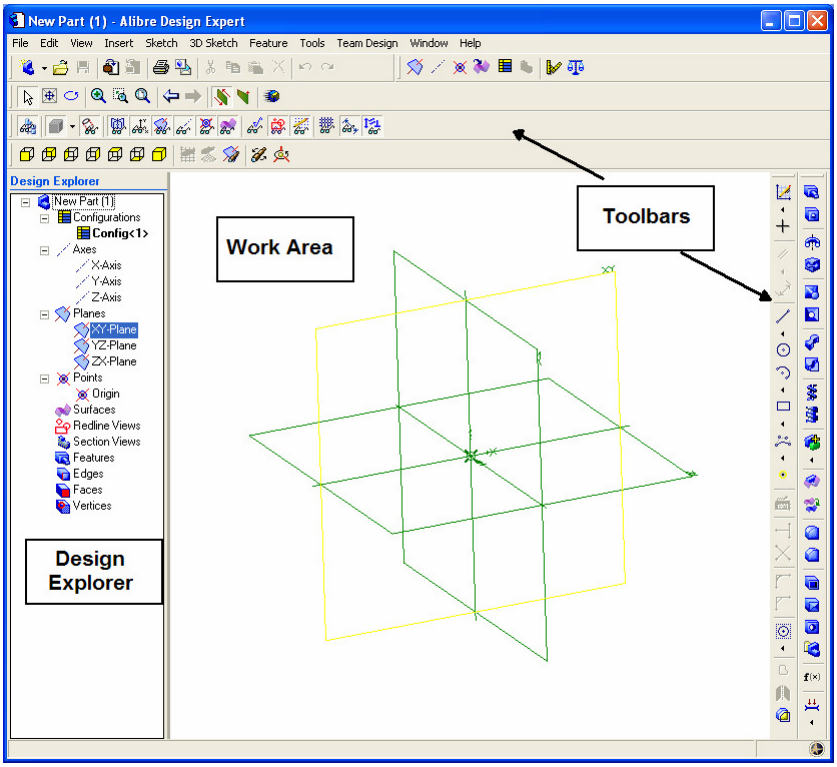
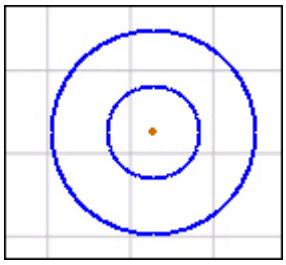


Figure 7: Part Workspace

Sketch Mode: You enter sketch mode in order to create a sketch that will later be used to create a 3D feature.

Sketch: A 2D profile made up of one or more entities such as lines, circles, and rectangles.



Sketch Constraints: Relationships applied to sketch geometry that controls their size, orientation, position or symmetry.

Feature: The result of a solid geometry operation that adds or removes material from a part, such as a boss, cut, hole, fillet, or chamfer.

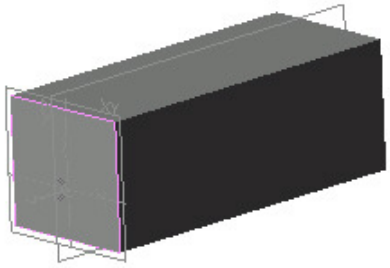
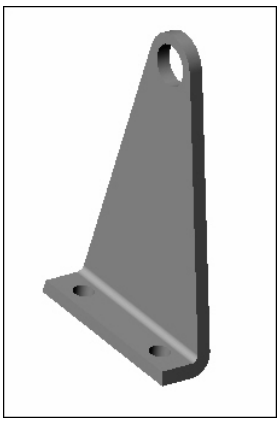
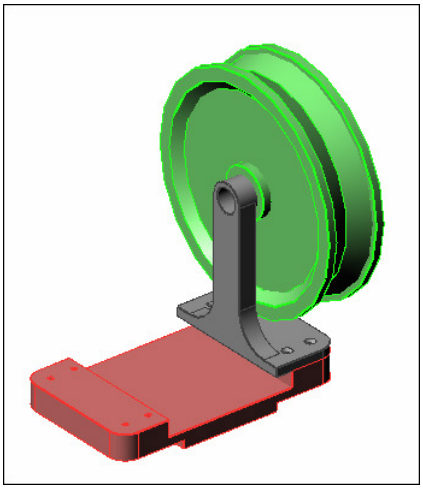


Figure 8: Extruded boss result

Part: A single 3d object that is comprised of one or more features.



Assembly: A model comprised of two or more parts and other assemblies (called subassemblies).



Assembly Constraints: Relationships that define the location and motion limitations of parts relative to other parts in an assembly.

Alibre Vault: A secure, complete Product Data Management (PDM) solution, available in some versions of Alibre Design.

2.6 Toolbars

Many of the commands you will utilize while modeling are found in the toolbars along the right side and top of the workspace. Each workspace has toolbars specific to the tasks performed in that workspace. In addition to the workspace specific toolbars, there are other general toolbars that are available in all workspaces.

To access the toolbars, from the **View** main menu, select **Toolbars**. Or, you can right click in the toolbar area of a workspace and select Toolbars. In the Toolbars dialog, check the toolbars you want to display, and uncheck the ones you do not want to display.

General Toolbars

- **Filters**: Turns on and off various selection filters to enable you to click on and select the desired entity more easily.

  Select Features

  Select Surfaces

  Select 2D Sketches

- **Inspection**: Insert reference geometry, use the measurement tool, calculate physical properties.
- **Orient View**: Orient the model to various pre-defined positions such as front, right, and top.
- **PDF Publishing**: Publish a model to a PDF file (availability depends on your license).
- **Redline**: Mark up a model with redline shapes or text.
- **Standard**: Basic operations such as file, save, import and export.
- **View**: Manipulate the view with functions such as pan, zoom, rotate.
- **Visibility**: Manipulate the display of reference geometry, and change the model between various shaded or wireframe modes.

Workspace Specific Toolbars

Part & Sheet Metal Part Workspaces

- **Part Modeling** (or Sheet Metal Part Modeling): Create common Part or Sheet Metal Part features.
- **Sketching**: Common sketching functions such as entity creation, trimming, and patterning.

Assembly Workspace

- **Assembly Modeling**: Common assembly functions such as inserting parts, inserting assembly constraints, moving and rotating parts.

Drawing Workspace

- **Detailing**: Common drawing functions such as inserting views, inserting notes, managing layers.

CHAPTER 3

Introduction to the Design Interface

Alibre Design is a parametric solid modeling system. Parametric solid modeling involves applying dimensions and other parameters (such as angle values and offset distances) to define a 2D profile, and ultimately the 3D shape, of an object. These dimensions and parameters can be changed at anytime to easily modify designs and alter the shape of a model.

To begin creating your model, you first create a sketch, which is a 2D profile. From this profile, a 3D feature is created. Examples of features include holes, fillets, cuts, and revolutions. You can add as many features as necessary to fully define the part. From there, multiple parts can be put together to create an assembly. Using a solid modeling tool to create assemblies shortens the length of the design process by giving you an early look at how your parts will fit together. You can correct small errors in the fit between parts without having to spend time and money on a prototype early in the process.

Using Alibre Design, you can also create 2D drawings and Bills of Material from either parts or assemblies. Alibre Design is fully associative, which means that the parametric capabilities propagate through the parts, assemblies, 2D drawings, and Bills of Material. This means that if you modify a parameter in one location, the models are updated everywhere that parameter is used.

You can also import parts, assemblies, and drawings from other CAD systems into Alibre Design.

In This Chapter

Workspaces 13
Selection Methods 17
Toolbars 19
View Manipulation 19
Getting Help 21
Keyboard Hot-Key Descriptions 22

3.1 Workspaces

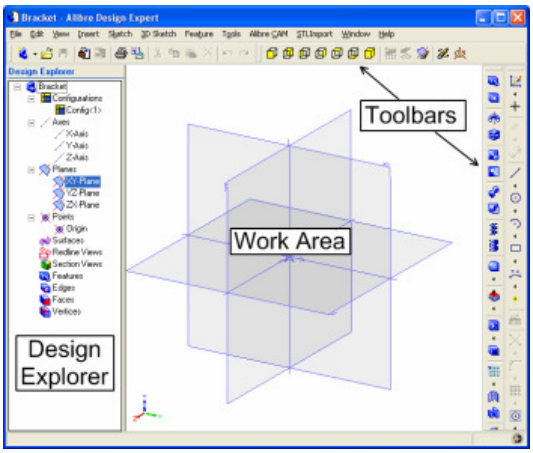
All design work in Alibre Design is done in windows called workspaces. You can open a part, sheet metal part (in Alibre Design Professional and Expert), assembly, drawing, or bill of materials workspace. Each workspace is displayed in a separate window; however, a drawing workspace can contain multiple drawing sheets. You can have as many workspaces open as needed.

3.1.1 Workspace Terms

Alibre Design workspaces are divided into two distinct areas.

- The **Design Explorer** is located on the left side of the workspace and lists all information related to a design including reference geometry and feature geometry.

- The **work area** is the section of the workspace in which you create all parts, assemblies and drawings. Toolbars may be located above and to the right of the work area.



3.1.2 Opening a New Workspace

➢ ***To open a new part, sheet metal part, assembly, drawing, or bill of materials workspace:***

Select the appropriate button on the Home Window.

Or,

Click the new **Part**, **Sheet Metal Part**, **Assembly**, **Drawing**, or **Bill of Materials** workspace tool in any open workspace.

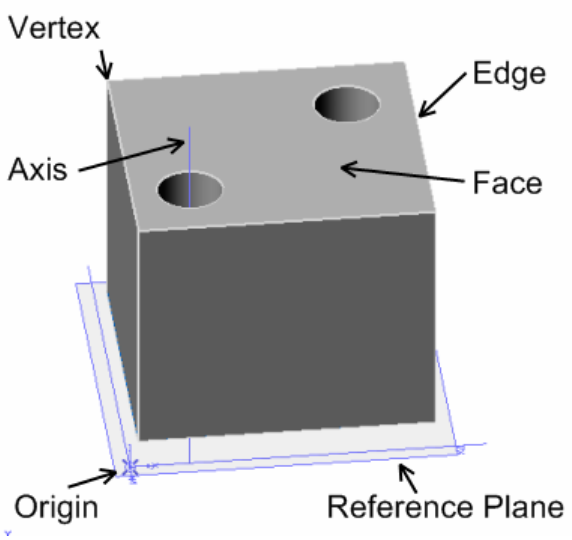
Or,

In the Home window, from the **File** menu, select **New**, and then select either **Part**, **Sheet Metal Part**, **Assembly**, **Drawing**, or **Bill of Materials**.

Note that sheet metal, BOMs, and drawings are not available in all versions of Alibre Design.

3.1.3 Model Terms

The following terms are consistently used throughout the documentation and refer to geometric elements in a model (faces, edges, and vertices), as well as construction geometry (reference planes, axes, and the origin).

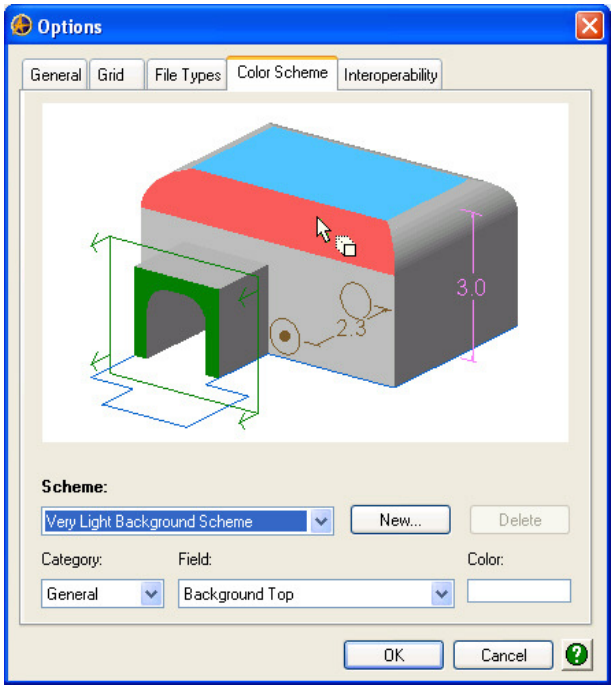


3.1.4 Work Area Color Scheme

You can change the color of the work area background and all other display components in a part or assembly workspace.

➢ ***To change the color scheme:***

1. In a part or assembly workspace, from the **Tools** menu, select **Options**.



2. Choose one of the four pre-defined color schemes, e.g. **Dark Background Scheme**, from the **Scheme** drop-down list, then click **OK** to apply the setting.

   Or,

Customize your display by creating a new color scheme.

➢ ***To create a new color scheme:***

1. Click **New**. The **New Color Scheme** dialog appears.

2. Enter a name for the new custom color scheme, then click **OK**.

Note: You cannot modify the default color schemes. If you attempt to change a color without creating a new color scheme, you will automatically be prompted to create a new scheme.

3. Select a category to modify from the **Category** drop down list. Next, select a **Field** in that Category. The diagram will show the current color scheme.

4. Click in the **Color** field to set a new color. Select the desired color, then select **OK** to apply the setting.

Note: A gradient background can be achieved by setting different colors for the 'General - Background Top' and 'General - Background Bottom' fields. A solid background can be achieved by setting these to be the same color.

5. Click **OK** to finish creating the new color scheme.

3.1.5 Multiple Views

You can split the work area into as many as four different views. You can zoom, rotate, and set the view mode in each view independently.

➢ ***To split the work area into multiple views:***

1. In a workspace, from the **Window** menu, select **Split View**.

2. Select **Horizontal**, **Vertical**, or **Both**. The Horizontal and Vertical options split the work area into two views. The Both option splits the work area into four views.

➢ ***To select a view and make it active:***

Click anywhere in the view border. A red arrow appears in the upper right corner of the active view.

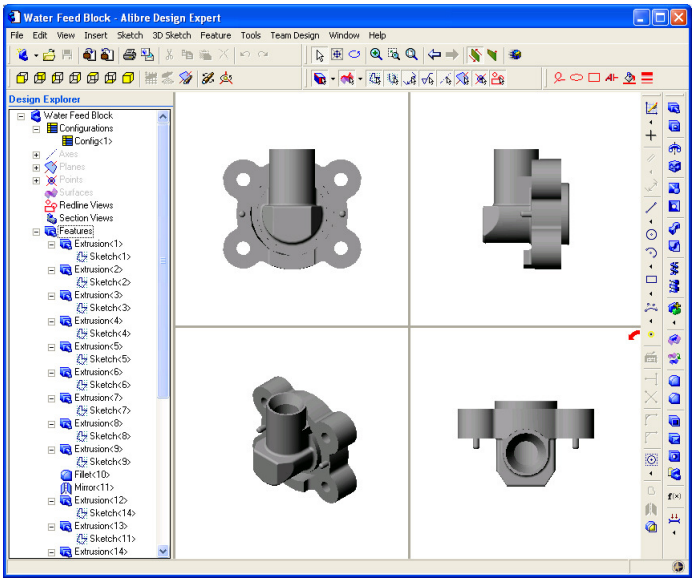
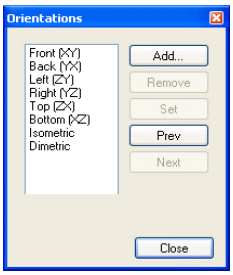


Figure 9: Workspace with Split Views

3.1.6 Named Views

In part and assembly workspaces, you can use named views to control view display and manipulation. You can quickly change the display to a default view and add custom views using the **Orientations** command under the **View** menu, or by selecting the **View Orientations** tool on the View toolbar.



To apply a named view to the work area, double-click a named view, or select a named view and click **Set**.

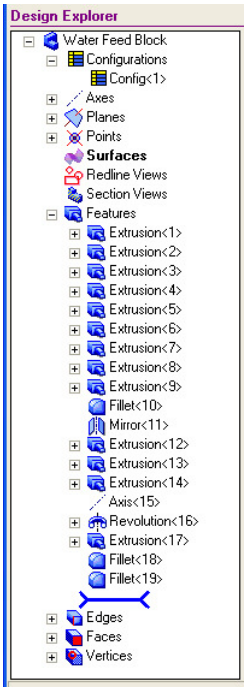
You can also add a custom view orientation by clicking **Add** and subsequently entering a view name.

3.1.7 Design Explorer

As previously described, each workspace consists of the work area and the Design Explorer. The Design Explorer's primary purpose is to track and list the structure of a part, assembly, or drawing. However, the Design Explorer can be used to accomplish numerous tasks.

Use the Design Explorer to:

- Select items in the design by name.
- Suppress or hide selected features and parts.
- Suppress or hide planes and axes.
- Temporarily roll the model or assembly back to an earlier state: double-click a feature or part in the Design Explorer or use the rollback bar.
- Identify and change the order in which features are regenerated.
- Rename features: right-click a feature and select **Rename**.
- Delete features and parts.
- Toggle the display of section views on and off.
- Edit sketches: right-click a sketch and select **Edit**.
- Track and control the display of redline markups.
- Check the status of a feature or part to resolve errors.



3.2 Selection Methods

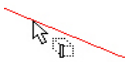
The majority of design tasks require that an item be selected. For example, a reference plane or planar face must be selected before you can begin sketching.

- **Selecting Individual Items** - To select faces, edges, vertices, etc., first click the Selection tool or choose **Select** from the **View** menu. Move the mouse pointer over an item, and then click once to select the item. An item will highlight as you move the mouse pointer over it and will change color after you select it.

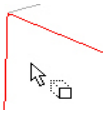
Note: Selection highlight colors will vary depending on the work area background color scheme used.

- **Selecting Multiple Items** - To select multiple items in the work area, hold the **Shift** key while you select the items. In cases where a multiple selection is required, you do not need to hold down the **Shift** key (except when using Advanced Selector). In the Design Explorer, holding the **Shift** key will select everything in between the items you selected. Hold the **Control** Key while selecting to choose only the items you click.
- **Selecting by Dragging -** In a sketch or a 2D drawing, select multiple items by dragging a selection rectangle around a group of items.
- **Using a Selection Filter** - Apply a filter to make selecting a specific item type easier. Filter on **Solid > Features**, **Faces**, **Edges**, and/or **Vertices**, in addition to **Surface > Surfaces, Faces, Edges,** and/or **Vertices**. To apply a selection filter, from the **Tools** menu, expand **Selection Filters** and choose the appropriate filter. You can also activate the **Selection Filters** toolbar and control filters easily with icons.
- **Face, Edge, and Vertex Cursors** - As you move the mouse pointer over faces, edges, and vertices, the mouse pointer will change to provide a visual indication of the item type.

Vertex Selection



Edge Selection



Face Selection

- **Modifying a Selection -** Many dialogs require a selection or multiple selections in order to complete the command. The selected item name will often populate an area in a dialog. To change the selection, either make another selection, which will override the previous selection, or right-click the item name in the dialog, and choose **Clear Selections** or **Remove Selected Item(s)**.
- **Advanced Selector** - On occasion, other items may obscure the item you want to select. Use the **Advanced Selector** to accurately select an item. To use the Advanced Selector, move the mouse pointer over the item, right-click it, and choose **Advanced Selector** from the pop-up menu. The **Select** dialog appears, containing all the items in the vicinity of the click location, including items hidden behind other faces. Select the item you want from the list.

Note: You can also **Ctrl-click** with the middle mouse button (if you have one) to access the Advanced Selector.

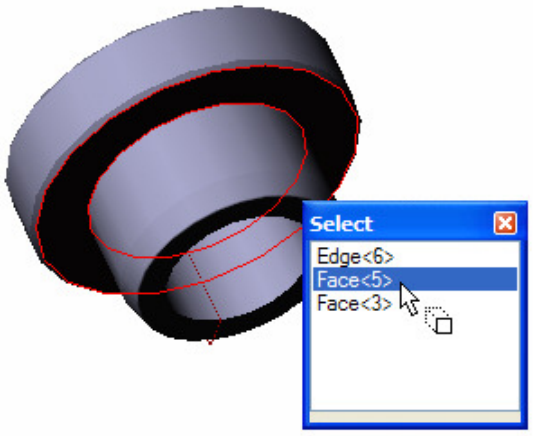


Figure 10: Advanced Selector Options

- **Selecting from the Design Explorer** - Select any item in the **Design Explorer** by clicking the item name. The associated item will change color in the work area. Select non-consecutive items in the Design Explorer by holding down the **Ctrl** key as you click. Select consecutive items in the Design Explorer by holding down the **Shift** key as you click.

3.3 Toolbars

You can control toolbar visibility, as well as toolbar position, in a workspace.

➢ ***To turn toolbars on and off:***

1. In any open workspace, from the **View** menu, select **Toolbars**; or, right-click in the toolbar area at the top of the workspace and select **Toolbars**. The Toolbars dialog appears.

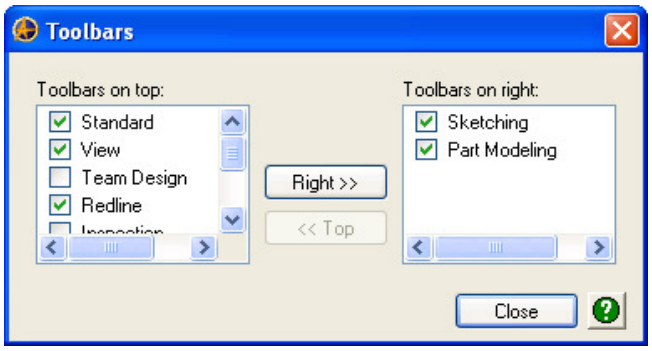


Figure 11: Toolbars Dialog

2. To hide a toolbar that is currently displayed, click the checked box next to the toolbar name.
3. To display a toolbar, click the empty checkbox next to the toolbar name.
4. Click **Close** to apply changes and exit the dialog.

Note: You can also turn individual toolbars on and off by right clicking in the toolbar area at the top of the workspace. A list of toolbars appears. Toolbars with checkmarks are currently on, and those without checkmarks are currently off. Toggle a toolbar on and off by clicking on it.

➢ ***To move a toolbar between the top and right of the workspace:***

1. In any open workspace, from the **View** menu, select **Toolbars**. The Toolbars dialog appears.
2. To move a toolbar from the top of the workspace to the right of the workspace, select the toolbar name and then click **Right**.
3. To move a toolbar from the right of the workspace to the top of the workspace, click the toolbar name and then click **Top**.
4. Click **Close** to apply changes and exit the dialog.

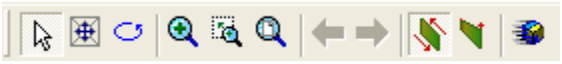
➢ ***To drag a toolbar to a different position in an area:***

1. Move the cursor near the vertical bar on the left side of the toolbar.
2. Click and hold the left mouse button and drag the toolbar to a new position within its respective area.

3.4 View Manipulation

You can change the way the model appears in the work area by modifying the orientation and display of the model. The tools available from the View toolbar, the Orient View toolbar, the Visibility toolbar, and the View menu allow you to manipulate the model in many different ways (rotating it, changing the zoom level, reorienting it, and viewing the model as a wireframe or shaded model, for example).

View Toolbar:



Orient View Toolbar:

Visibility Toolbar:

- The **Pan** tool dynamically moves the current view around the work area. Click the icon, and then click and drag the cursor around the work area.

Note: You can also pan by holding the **Shift** key and the left and right mouse buttons down simultaneously while moving the cursor around the work area. You can also pan by holding down the middle mouse button, if you have one.

- The **Rotate** tool rotates a part or assembly around a given point. Click the icon, and then click and drag the cursor around the work area.

Note: You can also rotate by holding the left and right mouse buttons down simultaneously while moving the cursor around the work area. If you rotate the model with the left and right mouse button method, you can switch dynamically between two types of rotation: mouse-centric or model-centric. To invoke mouse-centric rotation, click and hold the left and right mouse buttons while hovering over part of the model. If you click while hovering over white space, model-centric rotation will be used.

- The **Zoom Mode** tool dynamically changes the scale of the work area view. Click the icon, hold the left mouse button down, and move the cursor up to zoom in, or down to zoom out.

Note: If available, you can use the mouse wheel to dynamically zoom in and out.

- The **Zoom to Window** tool changes the zoom level of the view so that a specified region fills the work area. Select the tool, click and drag a rectangle around the desired area. Release the mouse button when the rectangle borders the correct area.
- The **Zoom to Fit** tool restores the view so that the entire design is displayed in the work area.
- The **Previous View** tool reorients the work area to views that preceded the current view.
- The **Next View** tool becomes available after the Previous View tool has been used.
- The **Orthographic** tool changes the display of the work area so that parallel edges, faces, etc. appear as infinitely parallel.
- The **Perspective** tool changes the display so that all parallel edges, faces, etc. appear to converge into one point.

- The **Display Acceleration** tool optimizes your display by simplifying the designs according to user-defined preferences. Refer to the section on *Display Acceleration* (on page 183) for more information.
- The **Orient** tools store default views of front, back, left, right, top, bottom, and isometric.
- The **Orient to Sketch Plane** tool changes the display so the sketch plane is displayed. This is only available in sketch mode. This is a toggle between the front and back view of the sketch plane.
- The **Isometric to Sketch Plane** tool changes the display so that it is an isometric view of the sketch plane.
- The **Orient to Plane** tool allows you to reorient the view to a plane of your choosing.
- The **View Orientations** tool stores default views (e.g. front, back, left, etc.), as well as custom views.
- The **Rotation Points** tool stores default rotation points (the origin, model's center of mass and center of volume), as well as custom rotation points.
- The **Model Shading** tools change the display to wireframe (so that only the edges of the model are shown, or shaded (so that the faces of the model are shaded).

- The **Toggle** tools allow you to toggle on and off the following options: Design Explorer, Silhouette Edges, All Reference Geometry, Coordinate System, Planes, Axes, Points, Surfaces, Annotations, Redlines, Sketches, Grid, Sketch Dimensions, and Constraint Symbols.

3.5 Getting Help

Help can be accessed numerous ways while you are using Alibre Design.

Integrated Help

- The integrated Help is automatically installed during installation of Alibre Design.
- To access Help related to a specific dialog, click the **Help** button in the dialog.
- When you mouse over an icon on a toolbar, a Tooltip will popup that identifies the function of the tool.
- The status bar in the lower left corner of a workspace displays hints related to completing a command and provides a brief description of a tool or function.

Tutorials

- Tutorials are available in the Help system under the Tutorials book.

Alibre Support website

You can obtain help from many areas on the Alibre, Inc. website. Go to www.alibre.com/support, and you will find links to:

- Frequently Asked Questions - a list that contains helpful information on a variety of topics. On the left side of the page, click the **FAQ** link.
- Knowledge Base - contains tips and instructions for performing specific tasks.

Interactive Support (for users with active maintenance agreements)

- If you are online, you can use Alibre's interactive, real-time support. An Alibre Application Engineer can help you, on your computer, while you watch and learn. Click the **Chat with Support** button

at the bottom of the Home Window to access this feature.
- You can contact the Alibre Application Engineers by phone or email during regular support hours.
- You can sign in to the Alibre Support web page and enter a support incident for the Alibre Application Engineers.

Note: Interactive support requests are handled in the order they are received. You should receive a response from an application engineer within approximately 24 hours.

3.6 Keyboard Hot-Key Descriptions

The following is a list of keyboard hot-keys for some of the most commonly used functions of Alibre Design.

General (for Part, Assembly, and Drawing Workspaces)

Open	CTRL + O
Select All	CTRL + A
Copy	CTRL + C
Cut	CTRL + X
Paste	CTRL + V
Save	CTRL + S
Save As	CTRL + SHIFT + S
Undo	CTRL + Z
Redo	CTRL + Y
Delete	DEL
Help	F1

Part Workspace

Design Properties	ALT + RETURN
Options	CTRL + SHIFT + O
Sketch Mode Toggle	CTRL + K
Orientation Dialog	CTRL + U

Measurement Tool Dialog	CTRL + M
Equation Editor Dialog	CTRL + E
Toggle Workspace	CTRL + TAB
Previous View	F3
Next View	F4
Regenerate	F5
Zoom to Fit	HOME
Toggle Sketches on/off	CTRL + SHIFT + K
Toggle All Reference Geometry on/off	CTRL + SHIFT + P

Assembly Workspace

Toggle Quick Constrain Mode	CTRL + Q
Assembly Properties	ALT + RETURN
Options	CTRL + SHIFT + O
Insert Part/SubAssembly	CTRL + SHIFT + I
Insert New Part	CTRL + SHIFT + N
Orientation Dialog	CTRL + U
Measurement Tool Dialog	CTRL + M
Equation Editor Dialog	CTRL + E
Toggle Workspace	CTRL + TAB
Regenerate	F5
Previous View	F3
Next View	F4

Drawing Workspace

Drawing Properties	ALT + RETURN
Options	CTRL + SHIFT + O
Sketch Mode Toggle	CTRL + K
Bill of Materials	CTRL + B
Print	CTRL + P
Toggle Workspace	CTRL + TAB

CHAPTER 4

Sketching

Sketching is fundamentally the most critical aspect of parametric solid modeling. The majority of features in Alibre Design begin with a sketch. A sketch is made up of one or more figures and provides the basic profile for a feature. Mastering sketching techniques is important and eventually leads to considerable time savings during modeling work.

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Customizing the Look of Sketch Figures 28
Sketch Figures 29
Reference Figures and Sketch Nodes 46
Working with Existing Sketch Figures 47
Sketch Constraints 54
Placing Dimensions on Sketch Figures 58
Working in a Sketch 66
Sketches and the Design Explorer 72

4.1 The Sketching Interface

The Sketching toolbar is shown by default on the right side of the workspace.

Activate Sketch (with options fly-out) ... activates sketch mode

Select ... select sketch figures and entities

Constraints (with options fly-out)... place manual constraints on a sketch

Dimension ... place dimensions on sketch figures

Line (with options fly-out) ... create a line figure (fly-out includes **Reference Line** ... create a reference line figure)

Circle (with options fly-out) ... create a circle figure (fly-out includes **Ellipse** ... create an elliptical figure)

Arc ... create an arc figure (options for circular and elliptical)

Rectangle (with options flyout)... create a rectangle figure (fly-out includes **Polygon** . . . create an n-sided polygon figure)

B-Spline (with options flyout)... create a spline figure

Sketch Node ... create a sketch node

Direct Coordinate Entry ... Creates sketch figures by entering point-to-point Cartesian or Polar coordinates

Trim (with options fly-out) ... delete part of a figure that extends beyond another one (fly-out includes **Extend** ... extend a figure up to another figure)

Fillet (with options fly-out) ... place a fillet on two existing figures (fly-out includes **Chamfer** ... place a chamfer on two existing figures)

Linear Sketch Repeat (with options fly-out) ... pattern a sketch figure in one or more linear directions (fly-out includes **Circular Sketch Repeat** ... pattern a sketch in a radial direction)

Sketch Shapes (with options fly-out)... create an individual or a pattern of a common shape

Offset ... offset an existing figure by a precise distance

Mirror ... create a symmetric copy of a figure about a reference

Project to Sketch ... use existing feature edges to create new sketch figures

Note: The **Extend**, **Trim**, **Fillet**, **Chamfer**, **Offset**, **Mirror**, and tools on the Sketching toolbar are only active when applicable sketch figures are in the sketch. For example, the **Extend** tool will not become active until at least two figures have been sketched.

The tools that are accessible on the Sketching toolbar are also accessible from the **Sketch** menu. The Sketch menu also contains tools that do not have a corresponding toolbar icon.

- **Ordinate Dimension** . . . Creates ordinate dimensions in 2D drawings.
- **Auto Dimension** . . . Automatically place dimension to place sketch.
- **Reference Figures** . . . Create reference figures of various shapes.
- **Move** . . . Move sketch figures from one location to another.
- **Rotate** . . . Rotate sketch figures about a center axis.
- **Analyze** . . . Determines if open ends, overlaps, or self-intersections exist in a sketch (also available on the Sketch Overlay).
- **Insert > Axis** . . . Inserts a 3D axis; sketch figures can be used as references.
- **Insert > Point** . . . Inserts a 3D point; sketch figures can be used as references.
- **Create Custom Symbol** . . . Creates a custom symbol using sketch figures.
- **Text > Field** . . . Inserts a text field into sketches.
- **Text > Label** . . . Inserts a text label into sketches.

4.2 Sketch Mode

Before creating a 3D feature, you must first create a 2D sketch, which serves as the profile (cross-section) for the feature. All 2D sketching is done in Sketch Mode. You must sketch on a reference plane or planar face. You will enter Sketch Mode to create new sketches or modify existing sketches.

4.2.1 Entering Sketch Mode

You must enter sketch mode before you can begin sketching.

➢ ***To enter sketch mode:***

Select the **Activate 2D Sketch** tool from the Sketching toolbar.

Or,

From the **Sketch** menu, select **Activate 2D Sketch**.

Or,

Right-click in the work area and select **Activate 2D Sketch** from the pop-up menu.

Or,

Press **Ctrl + K** on the keyboard.

You must sketch on an existing reference plane or face. You can select your sketch plane before or after entering sketch mode. If you did not select a sketch plane before you entered sketch mode, you will need to select a sketch plane after entering sketch mode, before you can begin sketching. If you have your cursor hints turned on (**Tools > Options > General** tab), you will be prompted by the cursor hint to select a sketch plane. You can select the sketch plane from the Design Explorer or in the work area. Once you select the sketch plane, all of the available sketching tools will become active. The Activate Sketch tool on the Sketching toolbar will always appear in the active state while in sketch mode.

4.2.2 Exiting Sketch Mode

➢ ***To exit sketch mode:***

You can use any of the methods listed for entering sketch mode by deselecting the chosen option, or use one of the following methods as well:

- Select the Exit button from the sketch mode overlay. Refer to *Sketch Mode Overlay* (on page 27) for details.
- Choose the **Select** tool from the View toolbar.
- Create a feature from the sketched profile. For example, select a feature tool such as the **Extrude Boss** tool from the Part Modeling toolbar.
- Select the **Regenerate** tool from the Part Modeling toolbar, or from the **Feature** menu, select **Regenerate All** or press the **F5** key on the keyboard.

4.2.3 Sketch Mode Overlay

When you enter sketch mode, an overlay will appear at the top of the work area. This overlay gives you quick access to grid options, the analyze sketch function, and the ability to create multiple sketches without leaving sketch mode.

You can turn the display of this overlay on and off by going to the **Tools** menu, and selecting **Options**. On the **Overlays** tab, you will see the following options:

- Show Full Overlay - Only applies to *Direct Editing* (on page 139) Functions
- Show overlay in 2D Sketch mode

To turn off the overlay in sketch mode, uncheck the option **Show overlay in 2D Sketch mode**. Check it to turn it back on.

When the overlay is turned on, you have the following capabilities:

- Check **Show Grid?** to display the sketch grid, and uncheck it to turn the display off.
- Set the grid **X Spacing** and **Y Spacing** as desired.
- Check **Snap to Grid?** to snap to the X & Y gridline intersections (you can snap to grid even if you have the grid display off).

Figure 12: Sketch Mode Overlay

- Select the **Analyze Sketch** tool to open the **Analyze Sketch** dialog.
- Select the **Refresh** tool to refresh the work area.
- Click **Apply** to accept the sketch you have created and begin a new sketch (you will be prompted to select a sketch plane to begin sketching again; you can choose the same sketch plane or a new one).
- Click **Exit** to accept the sketch you have created and exit sketch mode.

4.3 Customizing the Look of Sketch Figures

You can customize the colors of most object types, such as figures, reference figures, and nodes. You can also customize the size of nodes (endpoints of items, such as lines) to make them easier to view and select.

4.3.1 Customizing Colors of Items in Sketch Mode

To customize the colors of items found in Sketch Mode, you will need to set the colors of specific items on the **Color Schemes** tab of the Options menu.

➢ ***To change sketch colors***

1. Select **Tools > Options**.
2. Select the **Color Schemes** tab.
3. Under **Category**, select **Sketch**.
4. Set the colors of individual items shown in the **Field** drop down as you wish.
5. Click **OK**.

Note: If you change the color properties while you are in Sketch Mode, you may need to exit Sketch Mode and re-enter it for the changes to take effect.

4.3.2 Customizing the Size of Nodes

You may want to increase or decrease the size of sketch nodes. Nodes are basically the endpoints of sketch figures, such as the endpoints of a line or an arc. You will interact with nodes constantly, so it is important that you configure their size so that you can easily view and select them.

➢ ***To change the size of nodes***

1. Select **Tools > Options**.
2. Choose the **General** tab.
3. Under the Design Viewing area, select the size you want from the **Node Size** item.
4. Click **OK**.

Note: If you change the size of sketch nodes while you are in Sketch Mode, you may need to exit Sketch Mode and re-enter it for the changes to take effect.

4.4 Sketch Figures

You can create sketch figures while you are in Sketch Mode.

4.4.1 Line

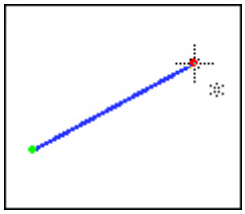
➢ ***To sketch a line:***

1. Select the **Line** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Line**; or right-click and select **Line** from the pop-up menu.
2. Position the cursor at the location you want to start the line.
3. Click in the Work Area to start the line and drag the cursor to sketch the line.
4. Click again to complete the line segment. You can continue to sketch additional line segments by clicking. Double-click or press **ESC** on the keyboard to end the line.

Note: During sketching, hints that provide step-by-step instructions are displayed in the lower left corner of the workspace.

➢ ***To resize a line:***

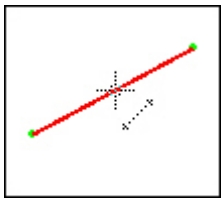
1. Select the **Select** tool from the Sketching toolbar.
2. Move the cursor over a node at the end of the line.
3. Click, hold the mouse button, and drag the node to resize the line.
4. Release the mouse button.



➢ ***To move a line:***

1. Select the **Select** tool from the Sketching toolbar.
2. Move the cursor over the line.
3. Click, hold the mouse button, and drag the line to a new location.

4. Release the mouse button.



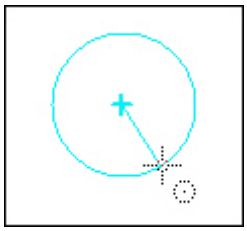
➢ ***To change the angle of a diagonal line:***

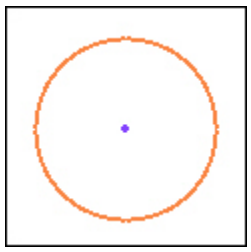
1. Select the **Select** tool from the Sketching toolbar.
2. Move the cursor over a node at the end of the line.
3. Click, hold the mouse button, and drag the node to change the angle.
4. Release the mouse button.

4.4.2 Circle

➢ ***To sketch a circle:***

5. Select the **Circle** tool from the Sketching toolbar; or from the **Sketch** menu, select **Figures > Circle**; or right-click and select **Circle** from the pop-up menu.
6. Position the mouse pointer at the center point location.
7. Click in the Work Area to place the center of the circle and release. Then, drag the mouse to sketch the circle.
8. Click again to complete the circle.





➢ ***To resize a circle:***

1. Click the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the circle.
3. Click, hold the mouse button, and drag the figure to resize the circle.
4. Release the mouse button.

➢ ***To move a circle:***

1. Click the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the center node of the circle.

3. Click, hold the mouse button, and drag the circle.

4. Release the mouse button.

4.4.3 Circular Arcs

You can sketch three different circular arc types: 1) **Center, Start, End**; 2) **Start, End, Radius**; 3) **Tangent-Start, End**.

➢ ***To sketch a circular arc using Center, Start, End:***

1. Select the **Circular Arc - Center, Start, End** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Center, Start, End**; or right-click and select **Circular Arc** from the pop-up menu.

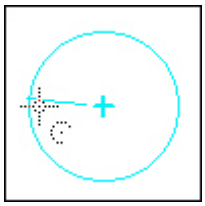
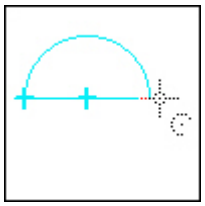
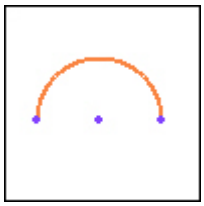
2. Position the mouse pointer at the center of the arc.

3. Click to place the center of the arc.

4. Click a second time to start the arc.

5. Move the mouse to sketch the arc.

6. Click a third time to complete the arc.

➢ ***To sketch a circular arc using Start, End, Radius:***

1. Select the **Circular Arc - Start, End, Radius** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Start, End, Radius**.

2. Position the mouse pointer at the arc starting location.

3. Click to start the arc.

4. Click a second time to place the arc endpoint.

5. Move the mouse pointer to size the arc.

6. Click a third time to complete the arc.

➢ ***To sketch a circular arc using Tangent-Start, End:***

1. Select the **Circular Arc - Tangent-Start, End** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Tangent-Start, End**.

2. Click the line or circular arc that will be tangent to the new arc. Click close to the side from which you want the new arc to originate.

3. Move the mouse pointer to size the arc.

4. Click a second time to complete the arc.

➢ ***To increase or decrease a circular arc's diameter:***

1. Click the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the circular arc.
3. Click, hold the mouse button, and drag the figure to resize the circular arc.
4. Release the mouse button.

➢ ***To reshape a circular arc:***

1. Click the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the circular arc center node.
3. Click, hold the mouse button, and drag the center node to reshape the circular arc.
4. Release the mouse button.

➢ ***To move a circular arc:***

1. Click the **Select** tool from the Sketching toolbar.
2. Select the arc AND center node by dragging a selection rectangle around the entities.
3. Hold the **Shift** key, click and hold the mouse button, and drag the circular arc.
4. Release the mouse button.

4.4.4 Rectangles

You can sketch two different rectangle types: 1) **Rectangle by Two Corners**; 2) **Rectangle by Three Corners**.

➢ ***To sketch a rectangle using Two Corners:***

1. Select the **Rectangle by Two Corners** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Rectangle > Two Corners**.
2. Click to place one corner of the rectangle.
3. Move the mouse pointer to sketch the rectangle.
4. Click a second time to locate the opposite corner of the rectangle.

➢ ***To sketch a rectangle using Three Corners:***

1. Select the **Rectangle by Three Corners** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Rectangle > Three Corners**.
2. Click to place one corner of the rectangle.
3. Move the mouse pointer and click a second time to locate the other corner on the same end of the rectangle.
4. Move the mouse pointer to adjust the rectangle length.
5. Click a third time to place the end of the rectangle.

➢ ***To resize a rectangle:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over an edge or a node.
3. Click, hold the mouse button, and drag the sketch entity to resize the rectangle.
4. Release the mouse button.

➢ ***To move a rectangle:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Select the entire rectangle by dragging a selection rectangle around it.
3. Hold the **Shift** key, click and hold the mouse button, and drag the rectangle.
4. Release the mouse button.

4.4.5 Spline Curves

Alibre Design provides NURBS (B-spline) curve functionality in sketches. The main distinguishing feature of the B-spline compared to "simple" splines is that it retains its shape at all times. That is, it can only translate and rotate rigidly in order to satisfy the constraints imposed on it. In addition to the B-spline curve, you can also create a spline curve by reference points. This spline curve does not retain its shape as a B-spline does.

Creation of spline curves by reference points

Splines created using this method are not B-splines. You specify a set of reference points in the work area to define the spline. You can use the Offset Figures tool on this spline, as described in the 'Using Offset on a Spline Curve' segment below.

1. From the **Sketch** menu, select **Figures > Spline > Create > Spline by reference points**.
2. Left-click to start the spline curve.
3. Move the mouse pointer and click a second time to place a reference point.
4. Move the mouse pointer to shape the curve.
5. Continue clicking to place additional reference points.

Note: You may choose to specify one or more reference points by invoking the direct coordinate entry tool and keying in the X, Y coordinates.

6. Double-click or hit **Escape** to complete the spline curve.

Note: By making the last reference point to be the same as the first reference point, a closed spline curve can be created.

➢ ***To modify a reference point spline curve:***

1. You can change the shape of a reference point spline curve by clicking on and dragging any of the reference points along the curve.

2. You can move the entire spline curve in the same way you move other sketch figures, by selecting the **Move** tool from the **Sketch** menu. You must select the entire curve by dragging a selection rectangle around it.

Creation of NURBS curves by control points

Using this method, you specify a set of control points in the work area to define the B-spline. Existing points can be identified. A preview is shown as you move the mouse.

1. Select the **B-spline by control points** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Spline > Create > B-spline by control points.**

2. Left-click to start the spline curve.

3. Move the mouse pointer and click a second time to place a control point.

4. Move the mouse pointer to shape the curve.

5. Continue clicking to place additional control points.

Note: You may choose to specify one or more control points by invoking the direct coordinate entry tool and keying in the X, Y coordinates.

6. Double-click or hit **Escape** to complete the spline curve.

Note: By making the last control point to be the same as the first control point, a closed B-spline curve can be created.

Creation of NURBS curves by interpolation

Using this method, you first specify a set of interpolation points in the Work Area that define the B-spline. A curve is then interpolated through the interpolation points.

1. Select the **B-spline by interpolation points** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Spline > Create > B-spline by interpolation points**.

2. Click to start the spline curve.

3. Move the mouse pointer and click a second time to place an interpolation point.

4. Move the mouse pointer to shape the curve.

5. Continue clicking to place additional interpolation points.

Note: One or more interpolation points may be specified via the direct coordinate entry tool.

6. Double-click or hit escape to complete the spline curve.

Constraining B-spline curves

The following constraints are supported on the B-spline curve:

- **Coincident constraint**: An existing reference point can be made coincident with a location on the B-spline by either dragging it onto the B-spline OR by using the sketch Coincident constraint tool.

Note: A point coincident to a B-spline is kept “floating," meaning that the constraint system can move the point of coincidence to any other location along the curve. However, any coincidence established at either endpoint of the B-spline remains fixed.

- **Tangent constraint**: can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.

Note: The note above on “floating” and “fixed” constraint applies to this constraint also.

- **Perpendicular constraint** can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.

Note: The note above on “floating” and “fixed” constraint applies to this constraint also.

- **Intersection point constraint** can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.
- **Fixed constraint** allows the B-spline curve to be locked in place.

Moving a B-spline curve

The main distinguishing feature of the B-spline is that it retains its shape at all times. That is, it can only translate and rotate rigidly in order to satisfy the constraints imposed on it. Other items to note include:

- Dragging a B-spline by its endpoint to existing sketch geometry introduces a “fixed” coincident constraint between the B-spline and that figure.
- Otherwise, dragging a B-spline to an existing reference point introduces a “floating” coincident constraint between the B-spline and that figure.
- Dragging an existing reference point to a B-spline introduces a coincident constraint between the point and the B-spline. It is “fixed” or “floating” depending on whether the point was dragged to the B-spline's endpoint or an internal location respectively.

➢ ***To move a B-spline curve rigidly:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Click and hold the mouse button on the curve, and drag the spline curve.
3. Release the mouse button to place the curve.

Trimming and extending B-spline curves

The B-spline curve can be trimmed using other figures (lines, arcs, circles, ellipses and B-splines).

Open figures like lines and arcs can be extended to the intersection point with a B-spline curve. However, the B-spline curve itself cannot be extended.

B-spline curve shape modification

An assortment of methods is available to modify or tweak the B-spline curve's shape ***while still honoring all the constraints placed on the curve***. These tools take advantage of the excellent local shape modification properties of B-splines.

➢ ***To modify a spline curve's shape:***

From the **Sketch** menu, select **Figures > Spline > Edit > Desired Action**

Possible Actions:

- **Move Control Points** - Modify shape by moving a control point
- **Move Curve Points** - Modify shape by moving a point on the B-spline curve to a new position.
- **Insert Knots** - Insert new knots on the curve knot vector without changing curve shape.
- **Remove Redundant Knots** - Remove existing knots that can be removed without changing the curve shape. Alibre Design will automatically remove any possible knots; then a dialog will appear summarizing the results.

DXF/DWG import/export of NURBS curves

NURBS geometry defined by control points present in DXF/DWG files can be read and precisely represented by the same mathematical representation in Alibre Design.

DWG files can also contain splines that are defined by interpolation points. Alibre Design will read these and convert them into B-spline curves. While for most parts these curves will appear very similar to what they look like in AutoCAD, their shapes may not exactly match. Also, AutoCAD allows users to specify “tolerance” while defining splines by interpolation. Alibre Design will assume this value is always zero.

Using Offset on a spline curve

The offset tool can be used to select either B-spline curves or reference point spline curves for creating an offset curve. For more information, see *Offsetting Figures* (on page 49). As with other offset figures, this offset will not be associative to the original curve. This means that if the original curve is modified, the offset figures will not update as a result. In addition, the offset curve will not contain any reference, control, or interpolation points.

4.4.6 Ellipses

➢ ***To sketch an ellipse:***

1. Select the **Ellipse** tool from the Circle fly-out on the Sketching toolbar; or, from the **Sketch** menu, select **Figures > Ellipse**.
2. Click to select the ellipse center.
3. Move the mouse pointer and click a second time to place the major axis of the ellipse.
4. Move the mouse pointer and click a third time to place the minor axis of the ellipse.

➢ ***To resize an ellipse:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the ellipse.
3. Click, hold the mouse button, and drag to reshape the ellipse.
4. Release the mouse button.

Note: You can also edit the dimension of the figure, if you have placed one.

➢ ***To move an ellipse:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Select the entire ellipse by dragging a selection rectangle around it.
3. Hold the **Shift** key, click and hold the mouse button, and drag the ellipse.
4. Release the mouse button.

4.4.7 Elliptical Arcs

➢ ***To sketch an elliptical arc:***

1. Select the **Elliptical Arc** tool from the Arc fly-out on the Sketching toolbar; or, from the **Sketch** menu, select **Figures > Elliptical Arc**.
2. Click to select the elliptical arc center.
3. Move the mouse pointer and click a second time to place the major axis of elliptical arc.
4. Move the mouse pointer and click a third time to place the minor axis of elliptical arc.
5. Click a fourth time to start the elliptical arc.
6. Click a fifth time to complete the elliptical arc.

➢ ***To resize an elliptical arc:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the elliptical arc or one of the associated nodes.
3. Click, hold the mouse button, and drag resize the elliptical arc.
4. Release the mouse button.

Note: You can also edit the dimensions of the figure, if you have placed one.

➢ ***To move an elliptical arc:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Select the entire spline curve by dragging a selection window around it.
3. Hold the **Shift** key, click and hold the mouse button, and drag the elliptical arc.
4. Release the mouse button.

4.4.8 Polygons

➢ ***To sketch an n-sided polygon:***

1. From the **Sketch** menu, select **Figures > Polygon**; or select the **Regular Polygon** tool from the Sketching toolbar. The Regular Polygon dialog appears.
2. Enter the number of sides for the polygon.
3. Choose to measure the internal or external diameter.
4. Left click once in the sketch window to locate the center of the polygon. Move the mouse to size the polygon. Left click again to place the polygon.
5. Choose **Apply**; then **Close**.

➢ ***To resize an n-sided polygon:***

1. Choose the **Select** tool from the Sketching toolbar.
2. Move the mouse pointer over the circle that defines either the internal or external diameter of the polygon.
3. Click, hold the mouse button, and drag to resize the polygon.
4. Release the mouse button

➢ ***To move an n-sided polygon:***

1. Click on the center node of the polygon and drag the polygon to a new location.

   Or,

   Choose the **Select** tool from the Sketching toolbar.
2. Select the entire polygon by dragging a selection rectangle around it.
3. Hold the **Shift** key, click and hold the mouse button, and drag the polygon.
4. Release the mouse button.

4.4.9 Sketch Shapes

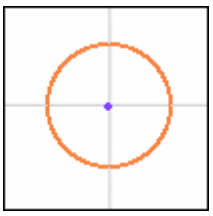
You can create a variety of standard shapes in Alibre Design. Each of these shapes can be placed alone or in a standard pattern.

Sketch Shapes are particularly useful for creating hole patterns with a large number of instances. Using the Sketch Shapes tool to create a pattern and then creating one extruded cut requires less regeneration time than creating a cut and then patterning the cut figure, for example.

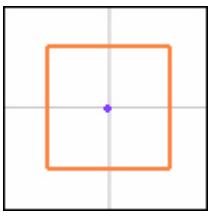
Each Sketch Shape or Sketch Shape pattern is treated as a single entity. Sketch Shapes can not be trimmed or extended, and other sketch entities can not be trimmed or extended to them. Sketch shapes can not be filleted or chamfered. Dimensions can only be applied to the nodes that are automatically placed with each sketch shape. (These nodes are typically placed in the center of each instance as well as the First Shape Anchor, if it is not in the center.)

Standard Shapes Available

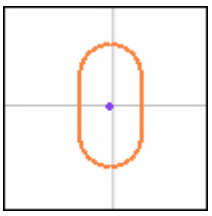
Round



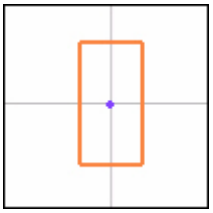
Square



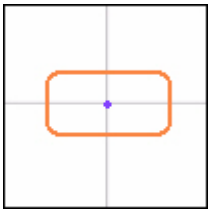
Obround



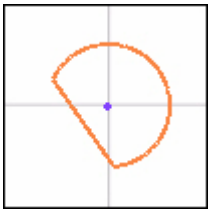
Rectangle



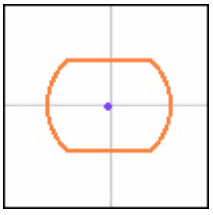
Rectangle with R Corner



Single D

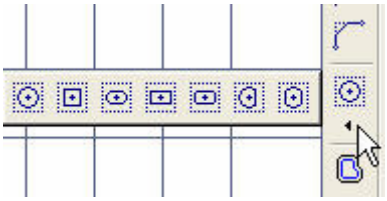


Double D

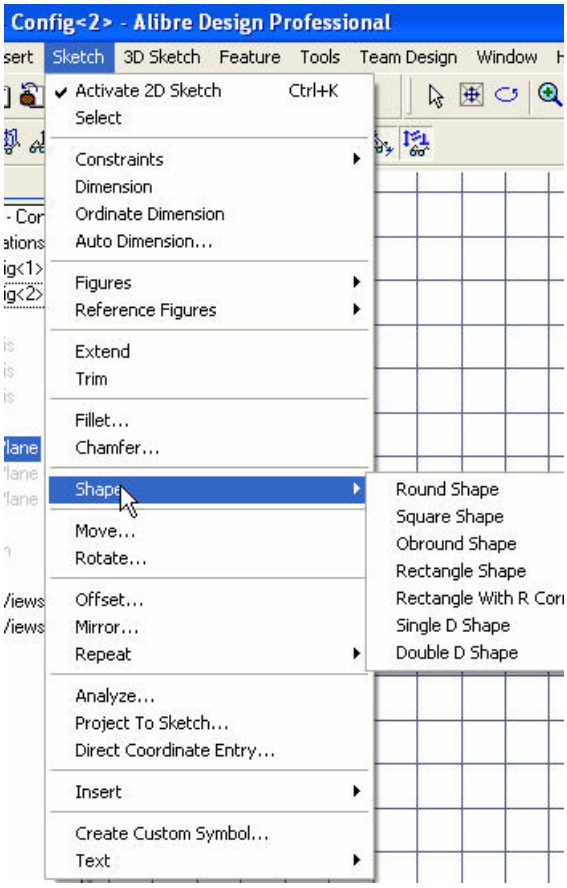


These shapes are available from the **Sketch** menu, as well as on the Sketching Toolbar.

- From the fly-out menu on the Sketching Toolbar:



- From the **Sketch** menu, select **Shape**:



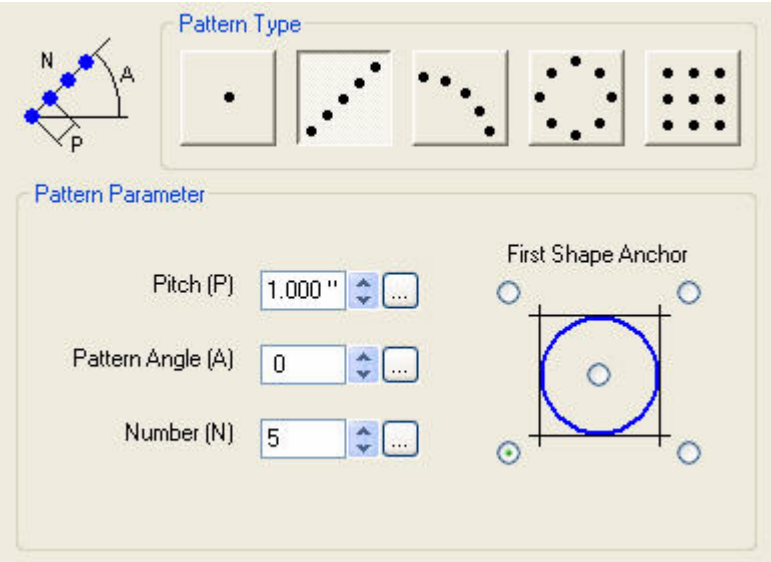
Standard Sketch Shape Pattern Types

The Pattern type field offers 5 different patterns to choose from.

No Pattern - creates a single instance of the selected shape.

Linear Pattern - creates a linear pattern of the selected shape.

When **Linear Pattern** is selected, the dialog options will look like this:

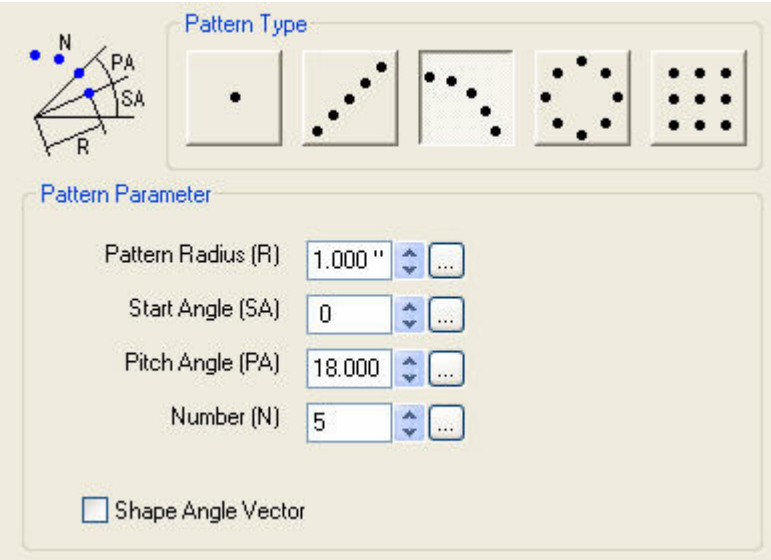


You will set the values for:

- Pitch - The distance between two successive shapes in the pattern
- Pattern Angle - The angle of inclination with respect to the positive X-axis
- Number - The number of shapes in the pattern (must be a value of 1 or greater)

Arc Pattern -creates an arc pattern of the selected shape.

When Arc Pattern is selected, the dialog options will look like this:



You will set the values for:

- Pattern Radius - The radius of the arc pattern
- Start Angle - The angle of the first shape of the pattern with respect to the positive X-axis
- Pitch Angle - The angle between two successive shapes in the pattern
- Number - The number of shapes in the pattern (must be a value of 1 or greater)

Check the box for Shape Angle Vector if you want the shapes in the pattern aligned along the radius of the arc pattern. (This option is not available for the round shape.)

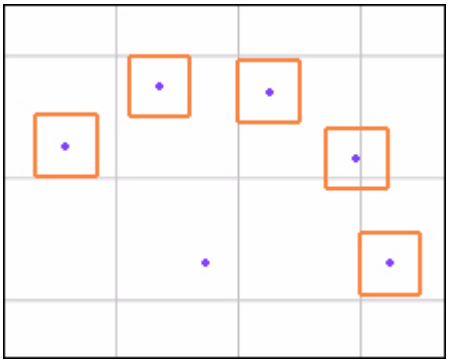


Figure 13: Shape Angle Vector Unchecked

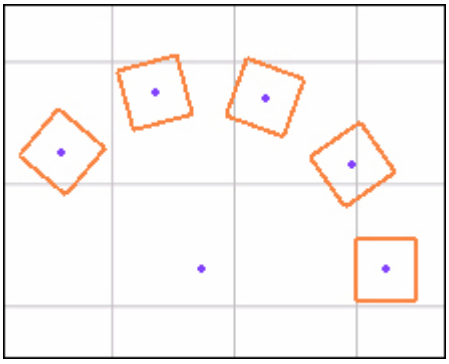
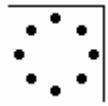
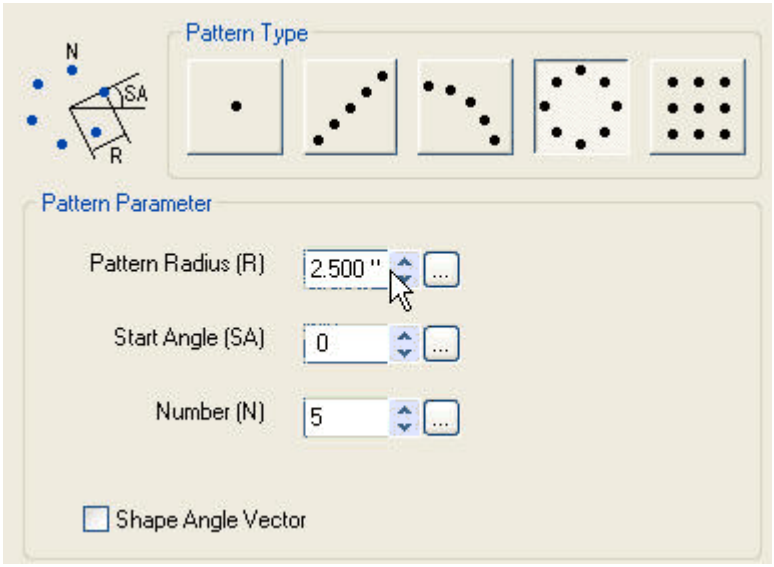


Figure 14: Shapes Angle Vector Checked

Circular Pattern - creates a circular pattern of the selected shape.



When Circular Pattern is selected, the dialog options will look like this:



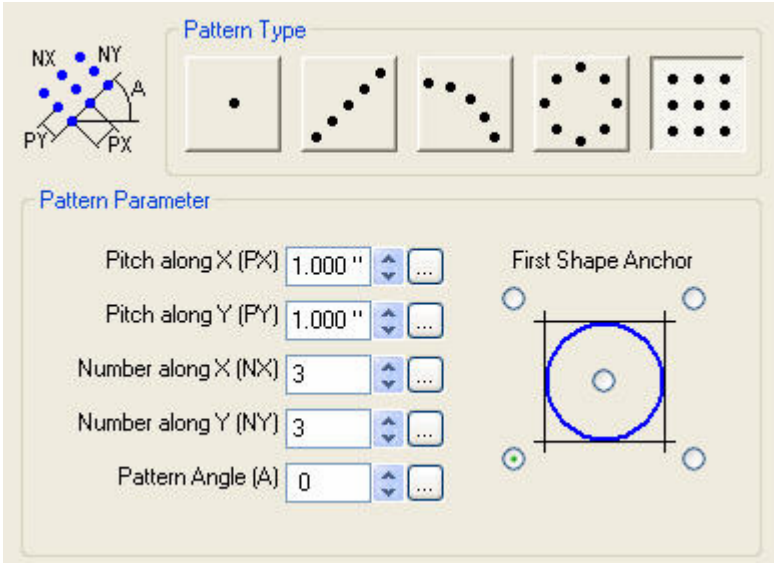
You will set the values for:

- Pattern Radius - The radius of the circular pattern
- Start Angle - The angle of the first shape of the pattern with respect to the positive X-axis
- Number - The number of shapes in the pattern (must be a value of 1 or greater)

Check the box for Shape Angle Vector if you want the shapes in the pattern aligned along the radius of the circular pattern. See Arc Pattern above for an example of how the Shape Angle Vector option works. (This option is not available for the round shape.)

Grid Pattern - creates a grid pattern of the selected shape.

When Grid Pattern is selected, the dialog options will look like this:



You will set the values for:

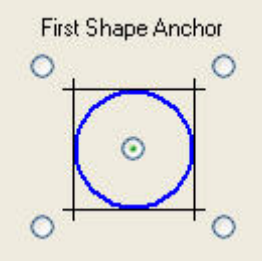
- Pitch along X - The distance between two successive shapes in the x-direction (noted in the diagram as PX)
- Pitch along Y - The distance between two successive shapes in the y-direction (noted in the diagram as PY)
- Number along X - The number of shapes in the pattern in the x-direction (noted in the diagram as NX; must be a value of 1 or greater)
- Number along Y - The number of shapes in the pattern in the y-direction (noted in the diagram as NY; must be a value of 1 or greater)
- Pattern Angle - The angle of inclination with respect to the positive X-axis

First Shape Anchor Location

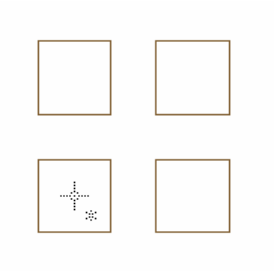
The first shape anchor location is where the mouse pointer is in relation to the first shape of the pattern when you are placing the shapes in the work area.

Examples of two of the available options are shown below. From these examples, you can see how the First Shape Anchor Location option works. This example pattern is a grid pattern of 2 shapes in the x-direction and 2 shapes in the y-direction, using the square shape.

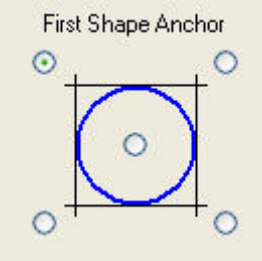
If you choose the center position...



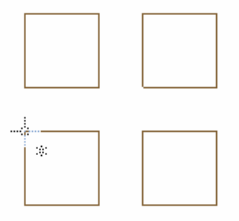
When placing the sketch shapes in the work area, the mouse pointer will be located in the center of the first shape. In the image below and all of the following images, the first shape is the square on the bottom left of the pattern.



If you choose the top left position...



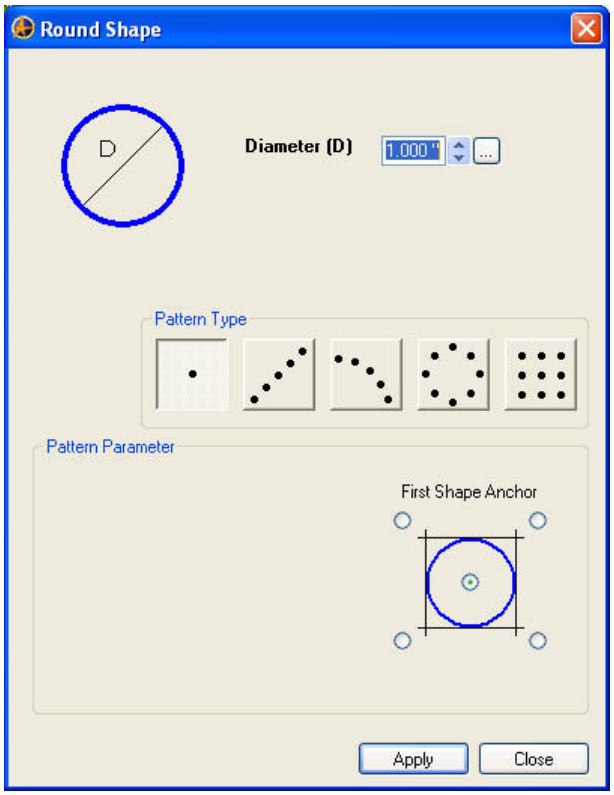
When placing the sketch shapes in the work area, the mouse pointer will be located at the top left of the first shape.



➢ ***To create a shape:***

1. Select the desired Shape tool from the Shapes fly-out .

   The Shape dialog for the chosen shape appears, and a preview of the figure is shown in the work area. This example shows the Round Shape Dialog.



2. Fill in the appropriate values for the chosen shape:

Round

In the Diameter [D] field, enter the desired value for the diameter.

Square

In the Side length along X field, enter the desired value for the side length of the square.

In the Angle field, enter the desired angle, measured from the x-axis.

Obround

In the Side length along X field, enter the desired value for the length of the shape.

In the Side length along Y field, enter the desired value for the width of the shape.

In the Angle field, enter the desired angle from the x-axis, if any.

Rectangle

In the Side length along X field, enter the desired value for the side length of the rectangle.

In the Side length along Y field, enter the desired value for the width of the rectangle.

In the Angle field, enter the desired angle, measured from the x-axis.

Rectangle with R corner

In the Side length along X field, enter the desired value for the side length of the rectangle.

In the Side length along Y field, enter the desired value for the width of the rectangle.

In the Radius field, enter the value for the radius of the corners.

In the Angle field, enter the desired angle, measured from the x-axis.

Single D

In the Side length along X field, enter the desired value for the side length of the single D.

In the Side length along Y field, enter the desired value for the width of the single D.

In the Angle field, enter the desired angle, if any, measured from the x-axis.

Double D

In the Side length along X field, enter the desired value for the side length of the double D.

In the Side length along Y field, enter the desired value for the width of the double D.

In the Angle field, enter the desired angle, if any, measured from the x-axis.

In the Pattern Type field, select the pattern type you desire.

3. In the **Pattern Parameter** field, enter the values that apply for the pattern type you selected.

4. Set the **First Shape Anchor** location (this option is not in Arc or Circular pattern types). This determines where the first shape in the pattern will be placed with respect to your mouse pointer when you click to place the shape.

5. Move your mouse pointer in the work area and click to place the figure. The Round Shape dialog remains open. You can continue to modify the values in the Round Shape dialog as needed.

6. Select **Apply** to accept the sketch figure. The sketch figure is placed, and centerpoints are shown for each shape. In addition, a preview for a new sketch figure appears with the mouse pointer. You can place another sketch shape pattern (you can modify any of the parameters when placing the next sketch pattern), or choose Close to exit the Round Shape dialog.

Note: Once a sketch shape pattern has been placed, all of the shapes in the pattern are grouped as one sketch figure. You cannot separate them. However, you can edit the shape pattern.

➢ ***To edit a shape or shape pattern:***

1. Right-click the shape in the work area and select **Edit**; or double-click the shape in the work area. The Round Shape dialog appears.

2. Make any changes necessary to the shape, then select **OK**. The figure updates to reflect the changes.

Note: The only way to edit sketch shapes is via the Shape dialog.

4.5 Reference Figures and Sketch Nodes

Reference figures and sketch nodes are used as construction geometry. Construction geometry is any entity that helps you model a part, but does not contribute to the physical properties of the model. For example, a reference line can be sketched and subsequently used in a sketch mirror operation. You can also place dimensions and constraints in relation to reference figures and sketch nodes. Reference figures and sketch nodes are contained within a sketch but are only visible in sketch mode. Reference figures are displayed as green dashed lines in sketch mode. Reference figures can be created in the exact same shapes as normal sketch figures.

➢ ***To sketch a reference figure:***

1. From the **Sketch** menu select **Reference Figures** > and then select a figure type.

2. To sketch the reference figure, follow the same steps you would use to create a normal sketch figure.

You can move and resize a reference figure just as you would a normal sketch figure.

➢ ***To place a sketch node:***

1. Select the **Sketch Node** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Node**.

2. Click once to place a sketch node.

➢ ***To convert a sketch figure to a reference figure:***

1. Right-click on a sketch figure.
2. Choose **Convert To Reference Figure**.

4.6 Working with Existing Sketch Figures

Several tools are included in Alibre Design to assist you in working with existing sketch figures.

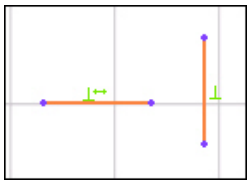
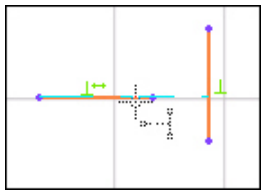
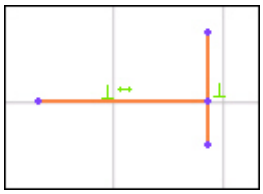
4.6.1 Extending Figures

You can use the **Extend** tool to extend a line or arc to meet another line, arc, circle, spline, or reference line.

➢ ***To extend a sketch figure:***

1. Select the **Extend** tool from the Trim/Extend fly-out on the Sketching toolbar; or from the **Sketch** menu select **Extend**; or right-click and select **Extend** from the pop-up menu.
2. Move the mouse pointer over the line or arc that you want to extend.

   A dashed preview will appear showing the direction of the extended entity. If the direction is incorrect, move the mouse pointer to the opposite end of the entity.
3. To generate the extension, click once on the entity.

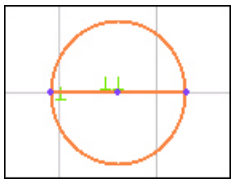
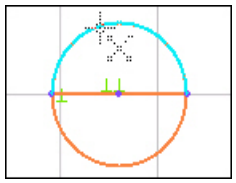
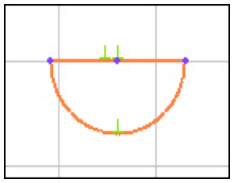




4.6.2 Trimming Figures

You can use the **Trim** tool to delete portions of sketch entities based on intersections with other entities. You can trim a line, arc, ellipse, circle, and spline that intersects with other lines, arcs, ellipses, circles, splines, and reference lines.

➢ ***To trim a sketch figure:***

1. Select the **Trim Figure** tool from the Sketching toolbar; or from the **Sketch** menu select **Trim**; or right-click and select **Trim** from the pop-up menu.
2. Move the cursor over the portion of the sketch figure that you want to trim. The portion becomes highlighted.
3. Click the highlighted portion to delete it up to its intersection with another sketch figure. The entire figure will be deleted if it does not intersect with another figure.

4.6.3 Adding 2D Fillets to Sketch Figures

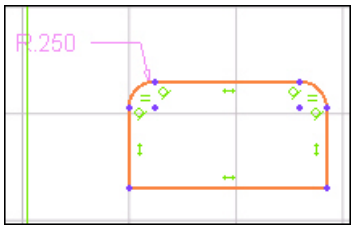
You can use the **2D Fillet** tool to place a tangent arc at the intersection of two sketch figures and subsequently delete the corner. You can also place a fillet on non-intersecting figures; the figures will be extended and a fillet will be placed accordingly at the resultant intersection.

➢ ***To add a 2D fillet to a sketch figure:***

1. Select the **2D Fillet** tool from the Sketching toolbar; or, from the **Sketch** menu select **Fillet**. The **Fillet Figures** dialog appears.
2. Select the first figure by clicking it. The figure name appears in the **Figures to Fillet** section of the dialog.
3. Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** section of the dialog.
4. Enter the fillet radius value in the **Radius** box.
5. Click **Apply** to create the fillet. (The Apply button will be inactive if the radius value is not possible.) The Fillet Figures dialog remains open so you can continue to place fillets on other figures.

Note: Consecutive fillets with a diameter equal to the first fillet will not be dimensioned; instead, an equal constraint will automatically be placed.

6. Click **Close** to close the Fillet Figures dialog.

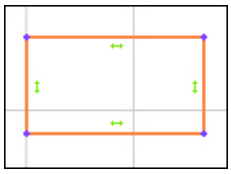


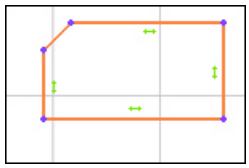
4.6.4 Adding Chamfers to Sketch Figures

You can use the **2D Chamfer** tool to place a beveled edge at the intersection of two sketch figures and subsequently delete the corner. You can also place a chamfer on non-intersecting figures; the figures will be extended and a chamfer will be placed accordingly at the resultant intersection.

➢ ***To add a 2D chamfer to a sketch figure:***

1. Select the **2D Chamfer** tool from the Sketching toolbar; or from the **Sketch** menu select **Chamfer**. The **Chamfer** dialog appears.
2. Select the first figure by clicking it. The figure name appears in the **Figures to Chamfer** section of the dialog.
3. Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** section of the dialog.
4. Enter the chamfer distance value in the **Distance** box.
5. Click **Apply** to create the chamfer. (The Apply button will be inactive if the distance value is too large.) The Chamfer dialog remains open so you can continue to place chamfers on other figures.
6. Click **Close** to close the Chamfer dialog.





4.6.5 Offsetting Figures

You can use the **Offset** tool to automatically create sketch figures offset from another selected figure or sketch by a specified distance.

➢ ***To offset a sketch figure or figures:***

1. Select the **Offset** tool from the Sketching toolbar. The **Offset** dialog appears.
2. Select the figure(s) to offset, either one at a time, or drag a selection rectangle around all figures. The figure name(s) appears in the **Figures to Offset** section of the dialog.
3. Enter the offset distance value in the **Distance** box.
4. If necessary, select the **Flip Direction** option to create the offset in the opposite direction.
5. Select a **Gap Type** (the default is **Natural).**

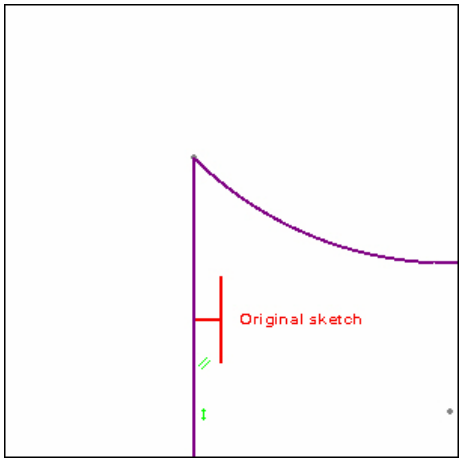


Figure 15: Original Sketch Example

Natural: Extends the edges of the wall along their natural curves until they intersect.

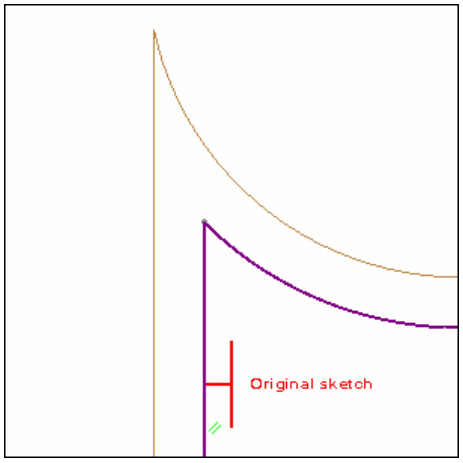


Figure 16: Result Using Natural Gap Type

Round: Creates fillets on any corners of the wall profile.

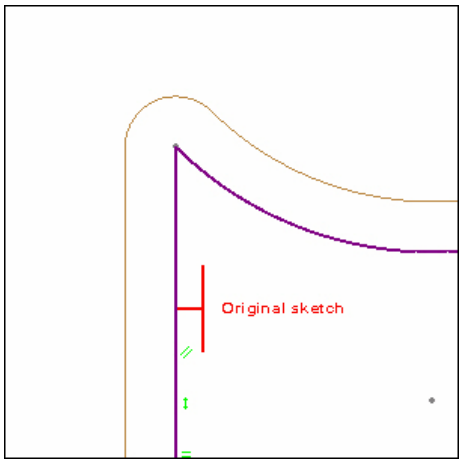


Figure 17: Result Using Round Gap Type

Extend: Extends the edges of the wall beyond their endpoints in straight lines until they intersect.

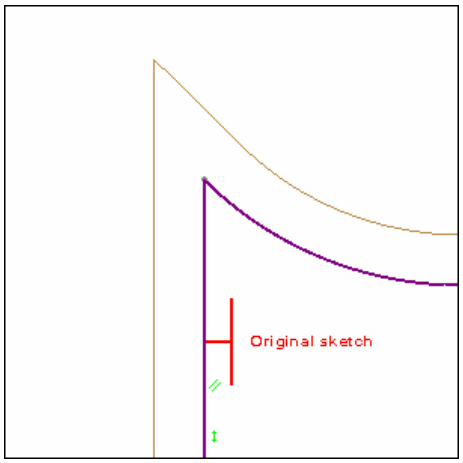
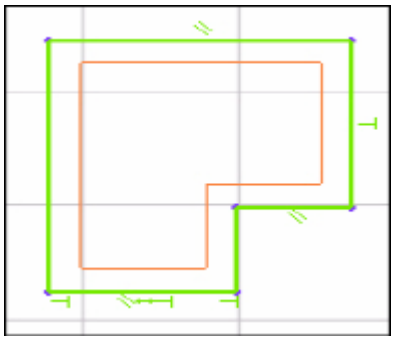


Figure 18: Result Using Extend Gap Type

6. Click **OK** to create the offset figure(s). The new figures are created and become part of the sketch.



4.6.6 Mirroring Figures

You can use the Mirror tool to create copies of figures mirrored about another reference line or figure. A **Symmetric** constraint is automatically applied between the original figure and the mirror figure. If you change the original figure, the mirrored figure will also change.

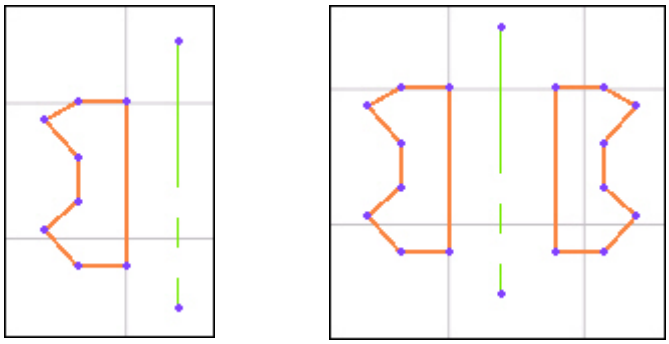
➢ ***To mirror a sketch figure:***

1. Select the **Mirror** tool from the Sketching toolbar. The **Mirror Figure** dialog appears.

2. Select the figure(s) to mirror either one at a time or drag a selection rectangle around all figures. The figure name(s) appears in the **Figures to mirror** section of the dialog.

3. Select the **Mirror Axis** to mirror the figure about. The mirror axis can be either a reference line or another sketch figure. You can NOT choose a reference geometry axis (such as the X-axis).

4. Click **OK** to create the mirrored figure.



4.6.7 Creating Patterns of Sketch Figures

You can create linear and radial patterns of an existing sketch figure. Linear patterns can be created in one or two directions.

➢ ***To create a linear pattern in one direction:***

1. Select the Linear Sketch Repeat tool from the Sketching toolbar; or, from the **Sketch** menu, select **Repeat > Linear**. The Linear Repeat dialog appears.

2. Select the figure to be patterned.

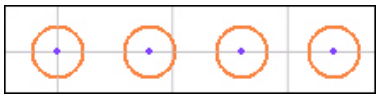
3. Select the linear path for the first pattern direction. Lines, reference lines, axes, and edges can be used as the linear path.

4. Enter the appropriate value in the **Copies** field; this value includes the original figure.

5. Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.

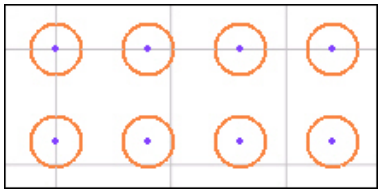
6. If necessary, select the **Change Direction** option.

7. Click **OK** to create the pattern.



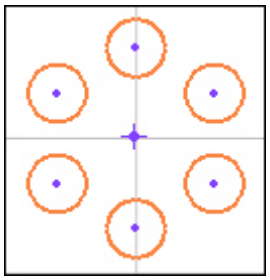
➢ ***To create a linear pattern in two directions:***

1. Select the Linear Sketch Repeat tool from the Sketching toolbar; or, from the **Sketch** menu, select **Repeat > Linear**. The Linear Repeat dialog appears.
2. Select the figure to be patterned.
3. Select the linear path for the first pattern direction. Lines, reference lines, axes, and edges can be used as the linear path.
4. Enter the appropriate value in the **Copies** field; this value includes the original figure.
5. Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.
6. If necessary, select the **Change Direction** option.
7. Select the linear path for the second pattern direction. The second linear path should not be parallel to the first.
8. Enter the appropriate value in the **Copies** field; this value includes the original figure.
9. Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.
10. If necessary, select the **Change Direction** option.
11. Click **OK** to create the pattern.



➢ ***To create a radial pattern:***

1. Select the **Circular Sketch Repeat** tool from the sketch repeat fly-out on the Sketching toolbar; or, from the **Sketch** menu, select **Repeat > Circular**. The **Circular Pattern** dialog appears.
2. Select the figure to be patterned.
3. Select the **Circular path center**. Axes, points, and existing edges on other features can be used as the circular path center.
4. Enter the appropriate value in the **Copies** field; this value includes the original figure.
5. Enter the angle that will separate each copy in the radial direction.
6. If necessary, select the **Change Direction** option.
7. Click **OK** to create the pattern.

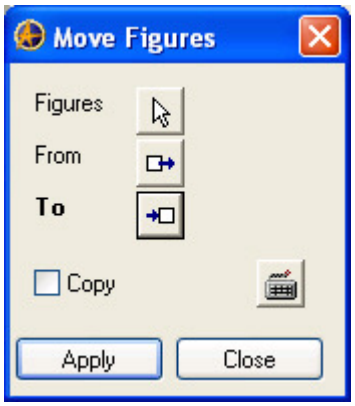


4.6.8 Moving and Rotating Sketch Figures

You can move existing sketch figures from one location to another, or rotate them about an axis.

➢ ***To move sketch figures:***

1. From the **Sketch** menu, select **Move**. The **Move Figures** dialog appears.



2. Click the **Figures** selection button. Select the figures you want to move in the work area.
3. Click the **From** button. Click the location in the work area that you want to move the figures from.
4. Click the **To** button. Click the location in the work area that you want to move the figures to.
5. Check **Copy** if you wan the original figures to be copied to the new location.
6. Click **Apply**; then **Close**.

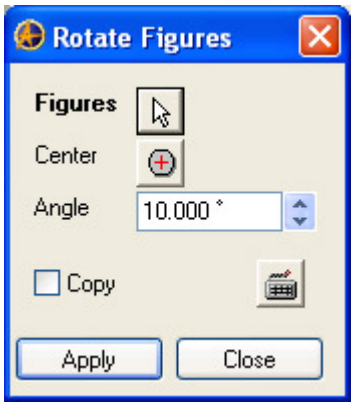
Note: If you want to move the figures using coordinates, click the **Direct Coordinate Entry** button.

➢ ***To drag sketch figures:***

1. Choose the sketch Select tool from the Sketching toolbar.
2. Select the figures to be moved. If you are moving an entire sketch, from the **Edit** menu, choose **Select All**, or press **Ctrl + A** on the keyboard. The figures become highlighted.
3. Hold the **Shift** key on the keyboard, and click and drag the sketch figure(s) to the new location.

➢ ***To rotate sketch figures:***

1. From the **Sketch** menu, select **Rotate**. The **Rotate Figures** dialog appears.



2. Click the **Figures** selection button. Select the figures you want to rotate in the work area.

3. Click the **Center** button. Click the location in the work area around which you want to rotate the figures.

Note: If you want to move the figures using precise coordinates, click the **Direct Coordinate Entry** button.

4. In **Angle**, set the angle of rotation.

5. Check **Copy** if you want the original figures to be copied to the new location.

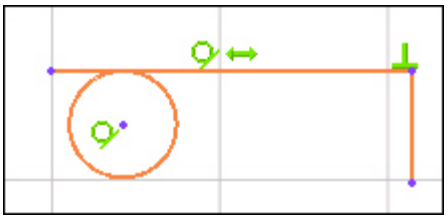
6. Click **Apply**; then **Close.**

4.7 Sketch Constraints

Figures in a sketch may be constrained to a size, orientation, and relationship to another 2D figure or 3D edge. Some constraints are used to apply relationships between figures (for example, perpendicular, tangent, parallel, equal size) or between figures and reference lines, planes, axes, vertices, and edges. Other constraints are applied to individual figures, including those that control a figure's orientation (for example, horizontal or vertical) and dimension.

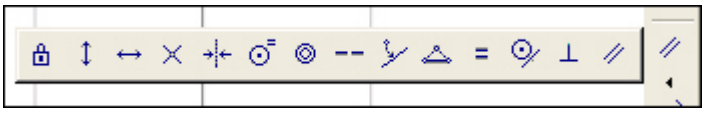
Some sketch constraints are applied automatically as figures are sketched. Horizontal, vertical, coincident, midpoint, tangent, intersection, and perpendicular constraint types are automatically placed depending on the type, size and orientation of the figure(s).

Sketch constraints are shown in proximity to the applicable sketch figure. In this example, the sketch constraints are shown enlarged.



Note: When you sketch a new figure on an existing figure or on a node on an existing figure (e.g. start a new line on the node at the end of an existing line), a **coincident** constraint is automatically applied. The coincident constraint is not displayed in this case. To break this coincident constraint, move the cursor over the coincident nodes, hold the **Ctrl** key, click and drag.

Constraints can also be applied manually to figures after they have been sketched with tools from the Constraints toolbar. The Constraints fly-out toolbar is available from the Sketching toolbar in part and drawing workspaces.



4.7.1 Constraint Types

Fourteen different constraints can be applied to sketch figures.

Fixed - Figures may be constrained to a fixed position in the sketch. After the constraint is applied, the node or figure may not be moved without first deleting the constraint.

Can be applied to: a node or any sketch figure

Vertical - One or more lines may be constrained to be vertical. Sketch nodes may also be constrained to be vertically aligned. Lines may be vertically constrained automatically as they are sketched or after placement.

Can be applied to: any line or any two nodes

Horizontal - One or more lines may be constrained to be horizontal. Sketch nodes may also be constrained to be horizontally aligned. Lines may be horizontally constrained automatically as they are sketched or after placement.

Can be applied to: any line or any two nodes

Intersection - Two figures may be constrained to intersect at a point.

Can be applied to: a point and any combination of arcs or lines

Symmetric - An axi-symmetric relationship may be defined between figures. After a symmetric constraint is applied, the figures are arranged axi-symmetrically and equidistant from a reference line or sketch line. The figures will become equal in size after the constraint has been placed.

Can be applied to: any two figures of like nature, e.g. two lines or two circles

Coradial - Figures may be constrained to share the same center point and same radius. Circles/arcs can be coradially constrained automatically during sketching or after placement.

Can be applied to: two or more arcs or circles

Concentric - Figures may be constrained to share the same center point. Circles/arcs can be concentrically constrained automatically during sketching or after placement.

Can be applied to: two or more arcs or circles

Collinear - Figures may be constrained so that they lie in the same line. Lines may be collinearly constrained automatically as they are sketched or after placement.

Can be applied to: a combination of two or more lines, axes, reference lines, edges

Coincident - A point can be constrained so that it lies on a figure.

Can be applied to: a point and any sketch figure

Midline - A node can be constrained so that it is fixed at the middle of a line. Midpoint constraints can be placed automatically as a figure is sketched or after a figure has been sketched.

Can be applied to: a node and a line or arc

Equal - Figures can be constrained to be equal in size. Equal constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: any two or more sketch figures

Tangent - Figures can be constrained to be tangent to a curve. Tangent constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: a curve and a line or two curves

Perpendicular - Lines can be constrained to be perpendicular to other linear entities. Perpendicular constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: two lines or a line and a circle or circular arc

Parallel - Lines can be constrained to be parallel to each other. Lines can be constrained parallel automatically during sketching or can be manually constrained after placement.

Can be applied to: at least two lines

4.7.2 Manually Applying Sketch Constraints

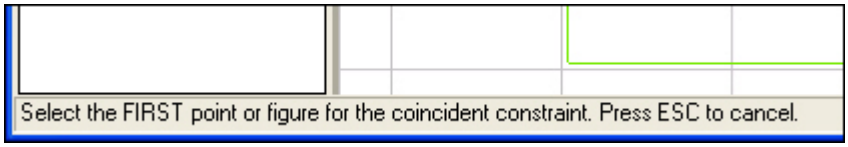
➢ ***To manually apply a sketch constraint:***

1. Click the small black options arrow on the Sketching toolbar to access the Constraint fly-out, or from the **Sketch menu**, select **Constraints**.

2. Select the applicable constraint tool. The mouse pointer changes to show the corresponding constraint symbol. For example, after selecting the **Coincident** constraint tool, the mouse pointer changes to .

3. In the sketch, select the figures to constrain. Many of the constraint types require multiple selections. For example, applying a symmetric constraint first requires selecting a reference line or sketch line, and then selecting two other sketch figures. Simply click all the required entities one-by-one to apply the constraint.

Note: When manually applying constraints, hints are displayed in the status bar in the lower left corner of the workspace. The hints provide step-by-step instructions to apply a constraint. You can turn these hints on and off by going to the **Tools** menu and selecting **Options**. Make sure **Status Hints** is checked on.



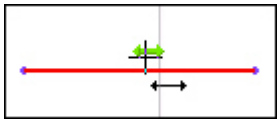
4.7.3 Deleting Constraints

Sketch constraints can be deleted at anytime, regardless of whether they were placed automatically or manually.

➢ ***To delete a sketch constraint:***

1. Select the sketch **Select** tool from the Sketching toolbar.

2. Position the mouse pointer over the sketch constraint you want to delete. The figure associated with the constraint is highlighted and the mouse pointer displays the selected constraint symbol.



3. When the constraint symbol appears on the mouse pointer, right-click the constraint and select **Delete** from the pop-up menu. The constraint is deleted.

Note: For constraints applied to multiple figures, deleting the constraint from one deletes that constraint from all the other figures in that constraint group.

4.7.4 Controlling the Display of Sketch Constraint Symbols

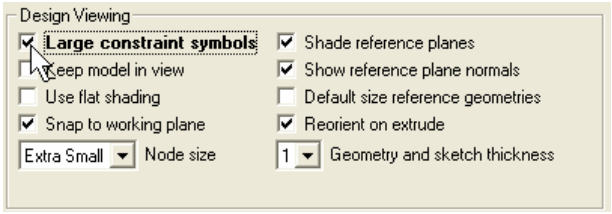
Sketch constraints symbol visibility can be turned on and off. In addition, you can control the size of the sketch constraint symbols.

➢ ***To turn off sketch constraint visibility:***

From the **View** menu, select **Constraint Symbols**, or press **Ctrl+Shift+C** on the keyboard. Figures will remain constrained when constraint symbols are hidden. This is a toggle on/off, so the visibility can be turned back on using the same command.

➢ ***To change the size of the constraint symbols:***

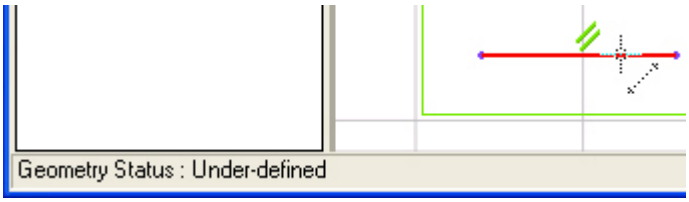
From the **Tools** menu, select **Options**. On the **General** tab, in the **Design Viewing** field, check **Large Constraint Symbols**. This will increase the size of the constraint symbols. Unchecking the option will return the symbols to their original size.



4.7.5 Checking the Status of a Sketch

Constraint status of an individual figure

There are several possible states that indicate whether a figure is constrained completely or not. A figure is constrained completely when zero degrees of freedom (the number of ways in which the sketch can still move within the sketch plane) remain. These states are displayed in the workspace status bar when a sketch constraint tool is selected and the cursor is positioned over a figure.



Well-defined: A figure is fully constrained and dimensioned; there are no remaining degrees of freedom.

Under-defined: A figure is not fully constrained or dimensioned; figures can move unexpectedly as a result.

Over-defined: A figure has conflicting constraints and/or dimensions that may or may not cause an additional constraint to fail. If a constraint fails, delete one or more constraints or dimensions.

Fixed: A figure is fully constrained and the figure cannot be modified. Other figures can be constrained to it.

Not-changed: The indicated constraint was not applied to this geometry. The figure is dependent on another figure with conflicting constraints.

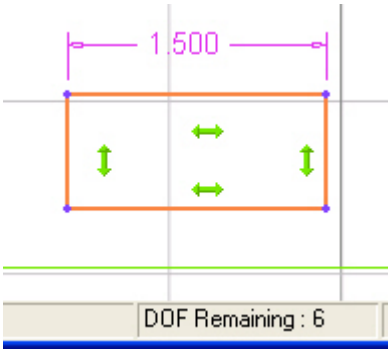
Not-consistent: The assigned dimension value(s) and constraints are in conflict and cannot be applied to the geometry.

Unknown: Occurs when a component of a constraint has been removed.

It is not necessary for figures in a sketch to be well-defined before you use the sketch to create a feature. However, it is good design practice in general to ensure that sketches are well-defined.

Status of entire sketch

By default, the number of remaining degrees of freedom (DOF) in the entire sketch is displayed in the status area in the lower right corner of a workspace. The DOF value will increase or decrease automatically as you sketch or delete figures, and add or delete dimensions and constraints. A fully defined sketch will have zero degrees of freedom. It is not required to fully define a sketch before it can be used in a feature operation.



➢ ***To hide the DOF hints:***

1. From the **Tools** menu, select **Options**. The Options dialog appears.
2. Select the **General** tab if it is not already selected.
3. In the **Hints** area, deselect **DOF** hints.
4. Click **OK**.

4.8 Placing Dimensions on Sketch Figures

Normally, to fully define and capture design intent in a sketch, you must place dimensions on sketch figures. However, it is not required to dimension sketches before they are used to create features. Most importantly, sketch dimensions can easily be changed and modified at any time. Additionally, any dimensions you place in a sketch will in turn be displayed in the 2D drawing that is based on the part when you choose to **Project Design Dimensions** when creating the drawing.

Two types of dimension states exist: driving and driven. Driving dimensions are used to define and constrain a figure. After driving dimensions have been placed on a figure, driven dimensions can also be added that are dependent upon the values of the driving dimensions. By default, driven dimensions are displayed in parentheses. Driven dimensions cannot be edited since they are dependent on driving dimensions. Subsequent changes to driving dimensions automatically update the driven dimensions.

In the figure below, the **2.000** and **.750** dimensions are driving dimensions, and the **1.250** dimension is a driven dimension (noted by parentheses). If the 2.000 or .750 dimension were changed, the 1.250 dimension would automatically update as well.

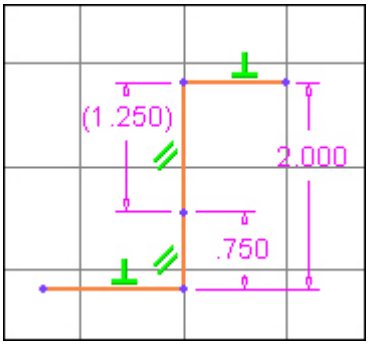


Figure 19: Sketch Figure Dimensions

4.8.1 Dimensioning Sketch Figures

Linear, radial, diametrical and angular dimensions may be created. Linear dimensions may be placed on a line, between two parallel lines, or between nodes. Diameter and radius dimensions may be created for circles and arcs. Angular dimensions may be created between two non-parallel lines.

➢ ***To dimension individual figures:***

Follow the steps below to place length dimensions on lines, and diameter dimensions on arcs and circles.

1. Select the **Dimension** tool from the Sketching toolbar; or from the **Sketch** menu, select **Dimension**; or right-click in the work area and select **Dimension** from the pop-up menu.

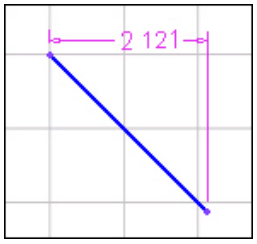
2. Move the cursor over the figure you want to dimension. The figure is highlighted.

3. Click the figure to show a preview of the dimension. Move the cursor to move the dimension preview.

Note: You can press the **Esc** key on the keyboard to cancel the current dimension operation.

Depending on the type of figure you are dimensioning, as well as where you move the preview dimension, a new dimension may be inferred. For example, depending on where the cursor is moved, three different dimensions could be placed on an angled line.

Figure 20: Actual Length



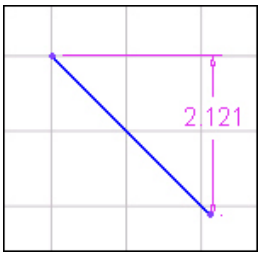
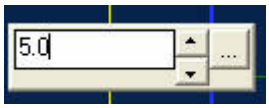


Figure 21: Vertical Distance

4. After the dimension has been positioned properly, click again. A dimension control box appears.



5. Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.

Note: You can enter fractions (e.g. **3/8**) and simple equations (e.g. **1.5 * 3**) into the dimension control box. You can also enter a value with units other than the current display units (e.g. **5 mm**). The value will be converted to the display units automatically. Supported unit abbreviations are **"**, **'**, **mm**, **cm**, and **m**.

➢ ***To dimension distances or angles between figures:***

Follow the steps below to place distance dimensions or angular dimensions between figures, e.g. between lines, between nodes, between arc or circle center nodes, etc.

1. Select the **Dimension** tool from the Sketching toolbar, or from the **Sketch** menu, select **Dimension**, or right-click in the work area and select **Dimension** from the pop-up menu.

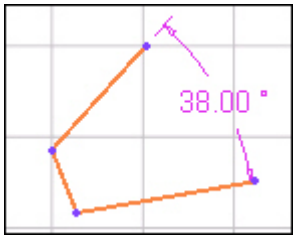
2. Move the cursor over the first figure from which you want to dimension.

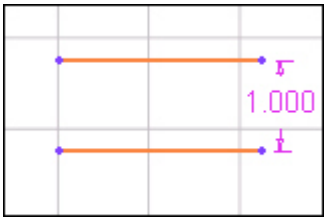
3. Click the figure. A preview may appear but do not place the dimension at this time.

4. Move the cursor over the second figure you want to dimension to and click again. A new dimension preview appears.

5. Move the cursor to position the dimension and click a third time. A dimension control box appears.

6. Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.





4.8.2 Auto Dimensioning a Sketch

You can automatically place driving dimensions on an entire sketch or on a selected subset of sketch figures. The number of dimensions placed automatically will vary depending on the number of sketch constraints that exist in the sketch. If you want to automatically place as many dimensions as possible initially, it is recommended you minimize the use of sketch constraints.

➢ ***To auto dimension the entire sketch:***

1. From the **Sketch**, select **Auto Dimension**. The Auto Dimension dialog appears.

2. Choose the option **All figures** and click the **Apply** button.

Note: The **Remaining DOF:** field shows you how many degrees of freedom you currently have remaining. You cannot modify this value.

3. Dimensions appear on the sketch. You can **Close** the dialog and modify any of the dimensions as required.

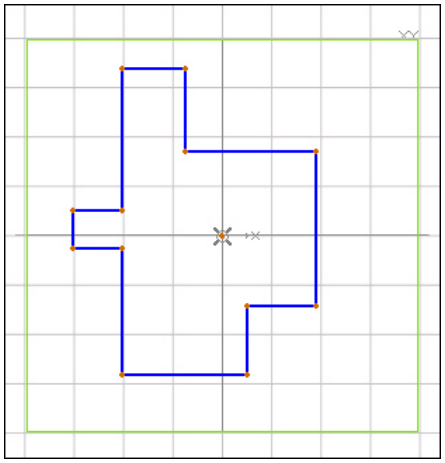


Figure 22: Before Applying Auto Dimension

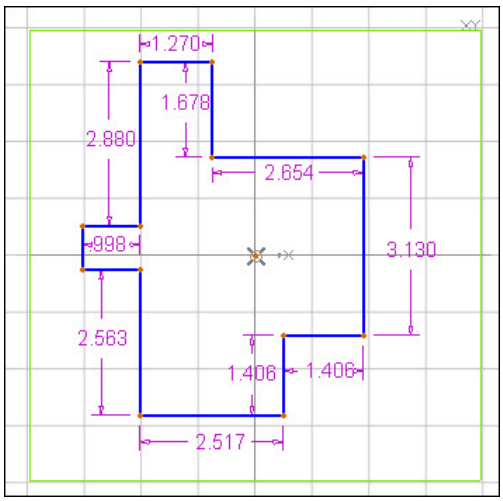


Figure 23: After Applying Auto Dimension

➢ ***To auto dimension a subset of the sketch figures:***

1. From the **Sketch**, select **Auto Dimension**. The Auto Dimension dialog appears.
2. Choose the option **Selected figures**.
3. Select one or more figures to dimension and click the **Apply** button.
4. Dimensions appear for the selected figures. You can either select and dimension additional figures or **Close** the dialog and modify the created dimensions.

4.8.3 Using Spinner Controls

When you place dimensions on sketch figures, you can enter the dimension value manually by typing a value. You can also use the spinner arrows to incrementally change a dimension value based on a pre-determined increment value.

➢ ***To set the spinner increment:***

1. From the **File** menu, select **Properties**. The Design Properties dialog appears.
2. Select the **Units** tab if it is not already selected.
3. In the **Spinner Increment** area, enter a **Length** increment value, i.e. .125”, .250”, .375”, etc.
4. Also enter an **Angle** increment value based on degrees.
5. Click **Apply**, and then click **Close**.

➢ ***To use the spinner arrows:***

1. Select the **Dimension** tool from the Sketching toolbar and dimension a figure.
2. When the dimension control box appears, click one of the two black arrow buttons to increase or decrease the dimension by the spinner increment value.

4.8.4 Using Equations in Dimensions

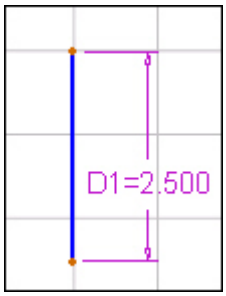
You can create dimensions using mathematical relations between dimensions or parameters, using dimension names as variables in the equations.

➢ ***To use equations in dimensions:***

You can type an equation in any text box where you would enter a dimension, or in the Equation Editor.

➢ ***To show equations in sketches and models:***

1. From the **File** menu, select **Properties**. The **Design Properties** dialog appears.
2. Select the **Dimension** tab.
3. Select the **Show Equations** option.
4. Click **Apply** and then click **Close**.
5. Place a dimension on a figure. The parameter name associated with the dimension is now displayed. If you included an equation, the equation will be displayed, but not the value.

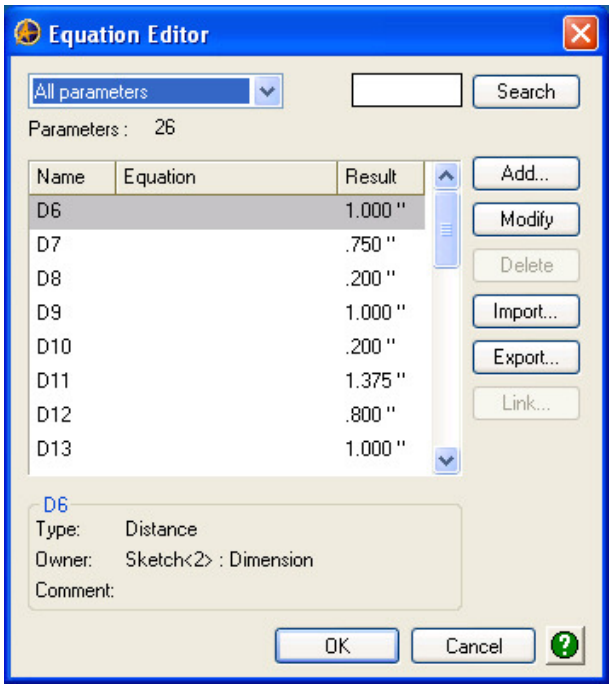


You can reference existing parameters as you create new dimensions. You can then manage the equations using the **Equation Editor**.

➢ ***To use the Equation Editor:***

1. You can access the **Equation Editor** directly when placing dimensions. Press the **Edit Equation** button on the dimension control box.

The **Equation Editor** dialog appears. In this dialog, you can add new equations and constants, use an existing equation to set the value of the dimension, or set the value equal to another dimension. The dimension parameters are listed under the **Name** column, the current values of the parameters are listed under the **Result** column, and the dimension type is listed under the **Type** column.



You can access the Equation Editor at anytime. From the **Tools** menu, select **Equation Editor**, or press **Ctrl + E** on the keyboard.

2. To modify an equation, select the equation from the list, and click **Modify,** or double-click in the field that you want to modify, i.e. name or equation.

3. If necessary, you can click **Add** to create new parameters. In the **Add Equation** dialog, specify the name of the new parameter, the type of parameter you want to create (Distance, Angle, Count, or Scalar), and the equation that will define its value. You can also add a comment if you desire. When you are finished, **Close** the **Add Equation** dialog. Type in the new variable name, new equation, or variable value and press **Enter** on the keyboard.

4. Click **OK** in the Equation Editor dialog to apply the changes.

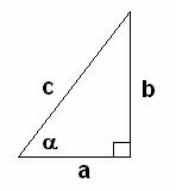
Functions available for use

The following functions are available for use in equations in Alibre Design.

Function	Name	Description
abs(x)	absolute value	Returns the absolute value of the argument.
acos(x)	arc cosine	acos (a/c) = a in radians |x| < 1
asin(x)	inverse sine	asin (b/c) = a in radians |x| < 1
atan(x)	arc tangent	atan(b/a) = α in radians |x| < 1
sin(x)	sine	Returns the sine of an angle. The argument can be any valid numeric expression in radians. The sin(x) function takes an angle and returns the ratio of two sides of a right triangle. The ratio is the length of the side opposite the angle divided by the length of the hypotenuse. To convert degrees to radians, multiply degrees by pi/180. To convert radians to degrees, multiply radians by 180/pi. sin(a) = b/c

Function	Name	Description
cos(x)	cosine	Returns the cosine of an angle. The argument can be any valid numeric expression in radians. cos(a)=a/c
tan(x)	tangent	The argument can be any valid numeric expression that expresses an angle in radians. tan(a)=b/a
int(x)	integer	Returns the integer portion of the argument. The argument can be any valid numeric expression. If the argument is negative, int(x) returns the first negative integer less than or equal to the number. If the argument is a positive decimal number less than 1, such as 0.885, null is returned.
frac(x)	fraction	Returns just the decimal portion of the argument.
sign(x)	sign	Returns the sign of the argument. The argument can be any valid numeric expression. If the number is greater than zero, sign(x) returns 1. If the number returns 0, and if negative, the sign(x) returns -1.
sqrt(x)	square root	Returns the square root of the argument: x > 0
X^n	X^n	x > 0

In the preceding table, “x” is a real number, “n” is an integer, and “a,” “b,” “c,” and “α” have the following relationship:



Dimensionality of equations

Each parameter has a specified dimensionality (that is, length, angle, scalar, count). When you write an equation for a parameter, the equation's dimensionality must match that of the parameter. If the equation's dimensionality is different, the equation will be displayed in red and a popup error message will appear when you rollover the equation.

For example:

- If D1 and D2 are the lengths of two line figures, you can write the equation, D2 = D1 * 0.50, which has the correct dimensionality (length), but you cannot write D2 = D1 * D1, because this equation has dimensionality “length squared”.
- If A1 is the angle between two line figures, you cannot write A1 = D1, because D1 is a length that will carry a unit, such as inches. The angle will be in degrees. These two do not have the same dimensionality.

4.8.5 Changing Sketch Figure Dimensions

➢ ***To change the value of an existing dimension:***

1. Select the sketch **Select** tool from the Sketching toolbar.
2. Move the cursor over the dimension. The dimension is highlighted.
3. Double-click the dimension. The dimension control box appears displaying the current dimension value.
4. Enter a new dimension value in the dimension and press **Enter** on the keyboard. The dimension is updated and the figure reflects the new dimension.

4.8.6 Deleting Sketch Figure Dimensions

➢ ***To delete an existing dimension:***

1. Select the sketch **Select** tool from the Sketching toolbar.
2. Move the cursor over the dimension. The dimension is highlighted.
3. Select the dimension by clicking it.
4. Press the **Delete** key on the keyboard.

   Or, right-click and select **Delete** from the pop-up menu.

   Or, from the **Edit** menu, select **Delete.**

4.8.7 Modifying Sketch Dimension Properties

You can change individual dimension properties such as dimension line size and style, dimension value format and precision, dimension text size and orientation, dual dimension display, and tolerance information.

➢ ***To modify sketch dimension properties:***

1. Select either the sketch **Select** tool or the **Dimension** tool from the Sketching toolbar.
2. Move the cursor over the dimension, right-click, and select **Properties** from the pop-up menu. The **Dimension Properties** dialog appears.

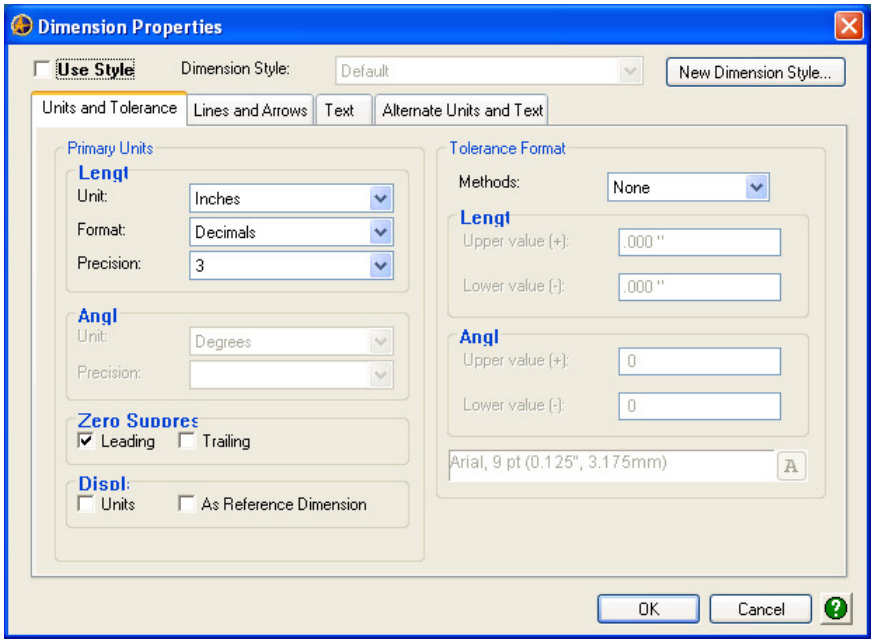


Figure 24: Dimension Properties Dialog

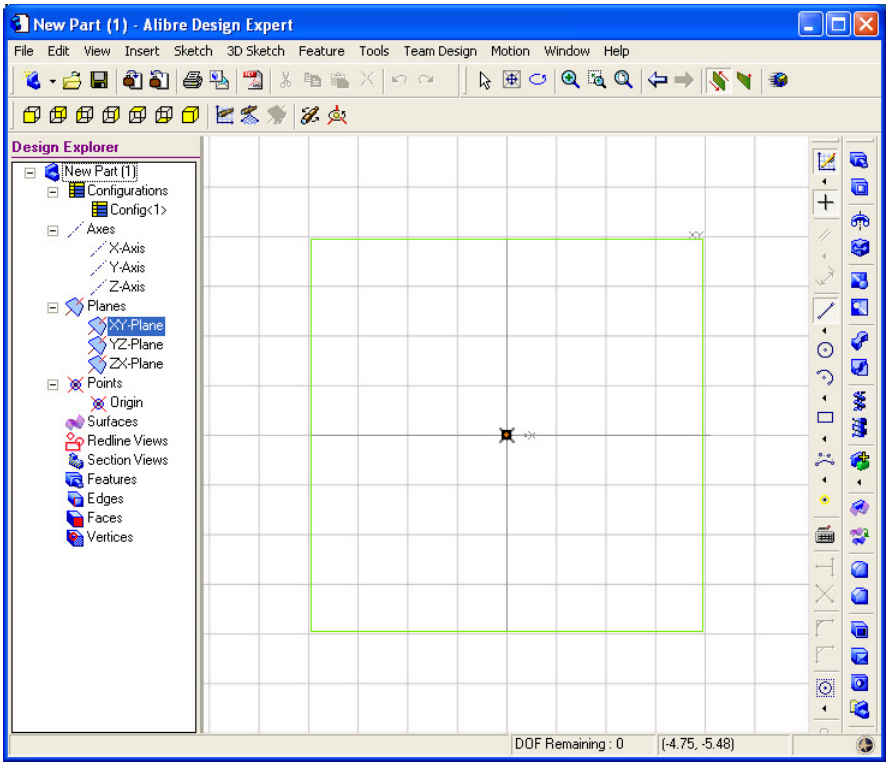
3. Select one of the four tabs: **Units and Tolerance**, **Lines and Arrows**, **Text**, and **Alternate Units and Text**.
4. Make the desired changes to the dimension settings.
5. Click OK to apply the changes.

4.9 Working in a Sketch

By default, a new part workspace contains three reference planes: **XY**, **YZ**, and **ZX**. Refer to *Reference Geometry* (on page 75) for details related to inserting additional reference planes). Any reference plane can be used as a sketch plane. To select a reference plane to be used as the sketch plane, simply select it either in the work area or the Design Explorer before entering sketch mode, or enter sketch mode and then select the desired sketch plane. You can refer to Entering Sketch Mode (on page 46) for more information.

4.9.1 The Sketch Grid

Upon entering a sketch, by default, a sketch grid will be displayed that can be used as reference during sketching.



The grid acts as an additional reference during sketching. You can customize the grid spacing, as well as choose to automatically snap to grid during sketching (snap to grid is not on by default). Both grid display and snap to grid are optional and can be turned on or off at any time.

➢ ***To turn the grid off:***

1. Uncheck the option **Show Grid?** on the Sketch Overlay (this is a toggle - check the box to turn it on).

   Or

   From the **Tools** menu, select **Options**. The Options dialog appears.

2. Select the **Grid** tab.

3. Uncheck the **Display grid** checkbox to turn the grid off (this is a toggle - check the box to turn it on).

4. Click **OK** in the Options dialog.

➢ ***To turn snap to grid on:***

1. Check the option **Snap to Grid?** on the Sketch Overlay (this is a toggle - uncheck the box to turn it off). If you cannot see the Snap to Grid? checkbox, you need to maximize your workspace.

   Or

   From the **Tools** menu, select **Options**. The Options dialog appears.

2. Select the **Grid** tab.

3. Check the **Snap to grid** checkbox to turn the snap to grid option on (this is a toggle - uncheck the box to turn it off).

4. Click **OK** in the Options dialog.

Note: You can snap to grid even if you do not have the grid displayed.

4.9.2 Snapping to the Working Plane

In sketch mode, you can have the work area view automatically reorient (once you have selected a sketch plane) so that you are looking directly at the sketch plane (the working plane).

➢ ***To turn Snap to the Working Plane off:***

1. From the **Tools** menu, select **Options**.

2. Select the **General** tab if it is not already selected.

3. In the **Design Viewing** area, deselect the **Snap to working plane** option to turn it on (uncheck it to turn it off).

4. Click **OK** in the Options dialog.

When this option is checked ON, the view will reorient once you have selected a sketch plane in sketch mode. When this option is checked OFF, the work area view will remain in its current orientation once you have selected a sketch plane.

➢ ***To reorient the view back to the sketch plane:***

While you are in sketch mode, you can reorient the view back to the sketch plane at any time.

From the **View** menu, select **Orient > To Sketch Plane**; or select the To Sketch Plane tool from the Orient View Toolbar .

➢ ***To reorient the view to the isometric of the sketch plane:***

While you are in sketch mode, you can reorient the view to the isometric of the sketch plane at any time.

From the **View** menu, select **Orient > Isometric To Sketch Plane**; or select the Isometric to Sketch Plane tool from the Orient View Toolbar .

4.9.3 Cursor Dimension Hints

As you sketch new figures, dimensional properties are, by default, displayed near the mouse pointer. For example, as you sketch a line, the line length and angle are displayed and updated automatically as you move the mouse pointer. You can hide the cursor dimension hints if desired.

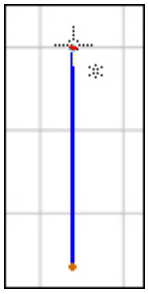
➢ ***To hide the cursor dimension hints:***

1. From the **Tools** menu, select **Options**.
2. Select the **General** tab if not already selected.
3. In the **Hints** area, deselect the **Cursor hints** option. (This turns off ALL cursor hints.)
4. Click **OK**.

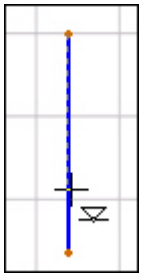
4.9.4 Cursor Display

For the most part during sketching, the mouse pointer's appearance will change depending on which sketch tool is selected, as well as the position of the mouse pointer.

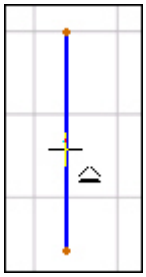
- The default symbol appears as when the sketch **Select** tool is selected.
- When another sketch tool is selected, the symbol will change to indicate the tool's function. For example, when the circle tool is selected, the symbol appears as .
- The mouse pointer also changes automatically depending on its position over an existing figure.



Node on Figure



On the Figure

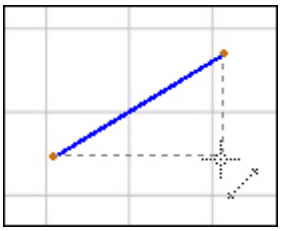


Midpoint of Figure

4.9.5 Inference Hints

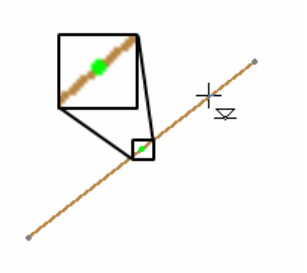
Inference Lines

During sketching, inference lines are displayed to provide a visual aid for aligning nodes. Inference lines appear as dashed lines and are automatically generated when your cursor is vertically or horizontally aligned with existing nodes or points, including the origin. You can change the color of inference lines in **Tools > Options > Color Schemes** by selecting **Inference Lines** under the **Sketch** category.



Midpoint Hints

While hovering over a line or arc, a midpoint hint will be displayed. Starting a figure on this hint will ensure the item starts on the midpoint.



4.9.6 Direct Coordinate Entry

You can enter Cartesian and/or Polar coordinates while sketching to define start/endpoints, center points, and angular and radial values.

➢ ***To use direct coordinate entry:***

1. Select any sketch figure tool.
2. Right-click in the work area and select **Direct Coordinate Entry** from the pop-up menu,

   OR, from the **Sketch** menu, select **Direct Coordinate Entry**.

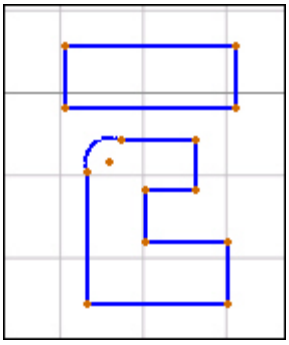
   OR, click the **Direct Coordinate Entry** tool from the toolbar. The **Direct Coordinate Entry** dialog appears.
3. Select either the **Cartesian** or **Polar** tabs.
4. Select either the **Absolute** or **Relative** options. The **Absolute** option will define all nodes with respect to the origin at (0,0,0). The **Relative** option will define all new nodes with respect to the last node entered.
5. Enter the Cartesian or Polar coordinates depending on which system is being used.
6. Click **Set** to define a node.
7. Click **Close** when finished.

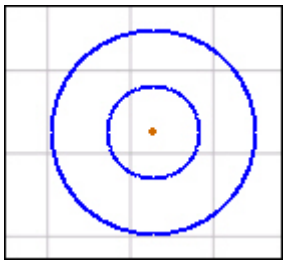
4.9.7 Right-click Menu

During sketching, a number of sketch tools are available from a menu that is quickly accessed by a right-click in the work area. Utilizing the right-click menu often provides the most efficient method in selecting a tool.

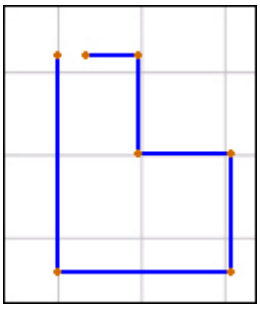
4.9.8 Open and Closed Sketches

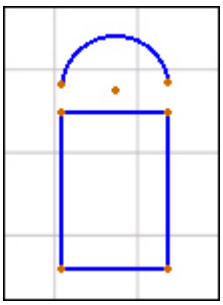
Sketches define the profile of a 3D feature. Most sketches will be closed, because the majority of 3D features require closed sketches. You can reference *Feature Creation* (on page 101) for more information related to features. A closed sketch contains no open-ended figures. The sketches below are examples of closed sketches.





An open sketch contains open-ended figures. Open sketches can only be used in **Thin Wall Boss** and **Thin Wall Cut** features. You can reference *Feature Creation* (on page 101) for more information related to features. The sketches illustrated below are examples of open sketches.





4.9.9 Checking Sketches for Errors

For complicated sketches involving many figures and nodes, it may be helpful to check for errors in the sketch such as open ends, intersections, and overlaps before creating a feature. If any of these errors exist, it may prevent you from successfully creating a 3D feature. Checking for these is also a valuable troubleshooting tool to resolve sketch problems.

Auto-Analyze Functionality

Alibre Design has an auto-analyze function that will analyze a sketch by default when you exit the sketch. You can turn the auto-analyze function off if desired.

When the auto-analyze function is turned on, Alibre Design will automatically run the Analyze Sketch tool each time you exit sketch mode, or when you choose a feature tool while inside a sketch. If the sketch has no errors, the process you have started will continue. If the sketch contains errors, you will receive a dialog alerting you to the error.

Choose one of the following options:

- **Show me** - this option brings up the Analyze Sketch dialog with the analysis results. Select any of the results and continue on with Steps 6-10 in the check for errors manually instructions that follow.
- **Ignore** - this option allows you to continue the process you started. Be aware that if you have started the process to create a 3D feature, the feature may fail as a result of the errors in the sketch.

Note: Check the Do not show me this screen option first, if you want to turn off the auto-analyze function. Alibre Design will not automatically analyze any future sketches unless you turn the option back on as described below.

➢ ***To turn on and off auto-analyze functionality:***

1. From the **Tools** menu, select **Options**.

2. On the **General** tab, in the Design Interaction section, uncheck the option for **Automatically Analyze Sketches** to turn it off (check the option to turn it back on).

➢ ***To check a sketch for errors manually:***

1. Select the sketch **Select** tool from the Sketching toolbar.

2. Select either the entire sketch by dragging a selection rectangle over the desired sketch figures, or select the appropriate sketch figures one at a time. The figures are highlighted after selection.

3. From the Sketch menu, select **Analyze**; or, from the Sketch Overlay, select the **Analyze** tool . The Analyze Sketch dialog appears.

4. In the **Analyze** section, check the items you would like to search for: Disjoint Ends, Open Loops, Overlaps, Self-Intersections, and/or Degenerate Figures.

5. Click the **Analyze** button. The sketch errors will appear in the **Results** area of the dialog.

6. Click the result you want to view, and the area will highlight in the part work area. In some instances, you can use the **Heal** option to resolve the sketch. The Heal option is typically available for Disjoint Ends, Open Loops, and Degenerate Figures.

7. Select the **Tolerance** option in the **Heal** area of the dialog.

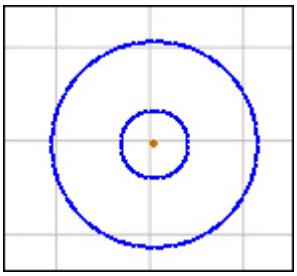
8. Enter a **Tolerance** value that is larger than the existing gap distance between the open nodes.

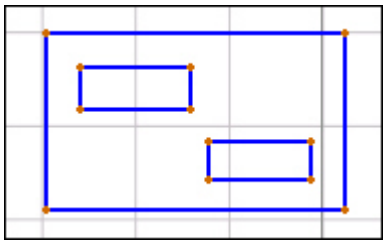
9. Once the Tolerance value is set, the **Heal** button will become active. Click the **Heal** button to resolve the figure.

10. Click **Close** to exit the dialog.

4.9.10 Enclosed Figures

A simple way to reduce steps when modeling a part is to create a sketch with enclosed figures. Enclosed figures are figures that are sketched within the profile of another figure. Material will be removed from the enclosed figure profile when a feature is created. The sketches illustrated below contain enclosed figures.





4.9.11 Copying and Pasting Sketch Figures

You can cut, copy, and paste entire sketches or individual sketch figures within the same sketch or into new sketches altogether.

➢ ***To copy and paste sketch figures within the same sketch:***

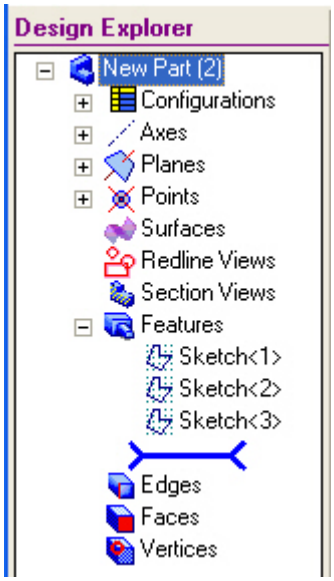
1. Select the sketch **Select** tool from the Sketching toolbar.
2. Select the figures to be copied. The figures become highlighted.
3. From the **Edit** menu, select **Copy,** or press **Ctrl + C** on the keyboard.
4. From the **Edit** menu, select **Paste,** or press **Ctrl + V** on the keyboard, or right-click and select **Paste** from the pop-up menu.
5. The copied figure(s) are placed slightly offset from the originating figures.

➢ ***To copy and paste sketch figures into a new sketch:***

1. Select the sketch **Select** tool from the Sketching toolbar.
2. Select the figures to be copied. If you are copying an entire sketch, from the **Edit** menu, select **Select All**, or press **Ctrl + A** on the keyboard. The figures become highlighted.
3. From the **Edit** menu, select **Copy,** or press **Ctrl + C** on the keyboard.
4. Exit sketch mode, select the new sketch plane, and enter sketch mode.
5. From the **Edit** menu, select **Paste,** or press **Ctrl + V** on the keyboard, or right-click and select **Paste** from the pop-up menu. The figures are pasted into the new sketch in the same orientation with respect to the origin as the originating sketch.

4.10 Sketches and the Design Explorer

After creating a sketch, and subsequently exiting sketch mode, the sketch will be listed in the Design Explorer under the **Features** node in the order it was created.



4.10.1 Editing Sketches

Sketches can be edited at anytime.

➢ ***To edit a sketch:***

1. You can get into edit mode for a sketch in one of the following ways:
    - In the Design Explorer, right-click the sketch name and select **Edit** from the pop-up menu.
    - In the Design Explorer, double-click the sketch name.
    - Double-click a sketch in the work area if it has not been used to create a feature.
    - If the sketch has been used to create a 3D feature, double-click the feature in the work area to edit the sketch used for that feature.
2. The sketch appears in sketch mode, and you can make the appropriate modifications to the sketch, and exit sketch mode to apply the changes.

4.10.2 Renaming Sketches

Sketches are listed in the Design Explorer by default as **Sketch<1>**, **Sketch<2>**, etc. You can rename the sketches to provide relevant information.

➢ ***To rename a sketch:***

1. In the Design Explorer, right-click the sketch name and select **Rename** from the pop-up menu, or click the sketch name twice with a slight pause between the first and second click. The sketch name is highlighted and can be changed.
2. Type in the new sketch name.
3. Press **Enter** on the keyboard.

4.10.3 Deleting Sketches

You can delete a sketch as long as it has not been used to create a 3D feature. To delete a sketch that has an associated feature, you must first delete the feature.

➢ ***To delete a sketch:***

In the Design Explorer, right-click the sketch name and select **Delete** from the pop-up menu, or select the sketch and press **Delete** on the keyboard.

CHAPTER 5

Reference Geometry

Reference geometry consists of planes, axes, points, and surfaces, which are primarily used for feature construction aids. Reference planes serve as the default sketch planes. Axes are fundamental to creating features such as revolutions and patterns. The primary reference point in a work area is the origin and is used extensively as a guide. Additional reference geometry can be added as necessary.

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Points 80
Reference Surfaces 81
Reference Geometry Visibility 84
Renaming Reference Geometry 84
Deleting Reference Geometry 85
Editing Reference Geometry Properties 85

5.1 Reference Planes

By default, three reference planes are visible in a part and assembly workspace, the **XY-plane**, **YZ-plane**, and **ZX-plane**; any can be used as the sketch plane. Additional reference planes can be inserted in any orientation and also used as the sketch plane. You can modify the display options of the reference planes.

➢ ***To modify plane display options:***

1. From the **Tools** menu, select **Options, General Tab**.

2. Check **Shade reference planes** to see the reference planes slightly shaded in the work area. Uncheck the box if you do not want to see the shading.

3. Check **Show reference plane normals** to see a 3D arrow designating the direction of a reference plane's normal when you place the cursor over it. Uncheck the box if you do not want to see the designating arrows.

5.1.1 Offset Plane

You can create a new reference plane parallel to an existing reference plane or planar face offset by a specified distance.

➢ ***To create an offset plane:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.

2. Select the existing reference plane or planar face to offset.

3. Enter the offset **Distance** value. (Use a value of 0.00 if you want the new plane to be on the existing plane or face.) A preview of the new plane is displayed.

4. If necessary, select **Reverse** to create the plane in the opposite direction.

5. Click **OK** to create the plane.

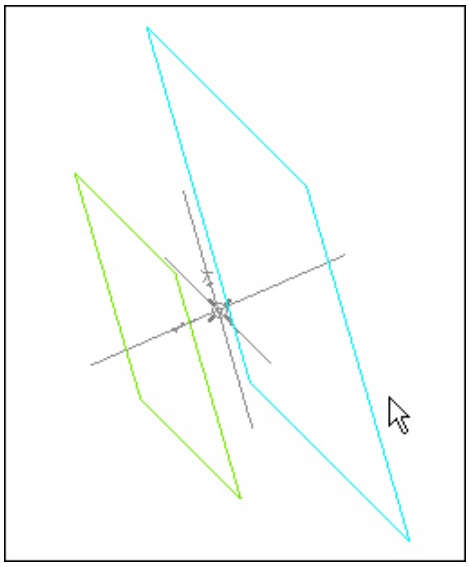


Figure 25: Offset Reference Plane

5.1.2 Tangent Plane

You can create a new reference plane parallel to an existing reference plane or planar face and tangent to an existing cylindrical face.

➢ ***To create a tangent plane:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.

2. Select the existing reference plane or planar face to offset.

3. Select the cylindrical face that the new plane will be tangent to. A preview of the new plane is displayed.

4. Check the **Reverse** option to choose the alternative tangent point.

5. If desired, you can select the **Symmetry Axis** option to create the plane parallel to the original plane or face and through the axis of the cylindrical face.

6. Click **OK** to create the plane.

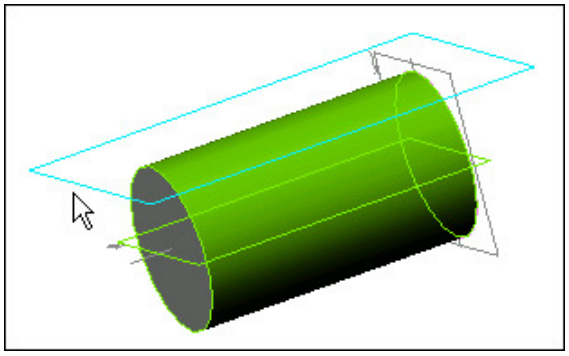


Figure 26: Tangent Reference Plane

5.1.3 Angled Plane

You can create a new plane through an edge or axis at an angle to an existing reference plane or planar face.

➢ ***To create an angled plane:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.
2. Select an existing edge or axis.
3. Select an existing reference plane or planar face. A preview of the new plane is displayed.
4. Enter the **Angle** value.
5. If necessary, select **Reverse** to create the plane in the opposite direction.
6. Click **OK** to create the plane.

5.1.4 Parallel Plane through a Point

You can create a plane parallel to an existing reference plane or planar face through an existing point.

➢ ***To create a parallel plane through a point:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.
2. Select an existing reference plane or planar face.
3. Select an existing reference point. A preview of the new plane is displayed.
4. Click **OK** to create the plane.

5.1.5 Plane at Line and Point

You can create a plane through an edge or axis and a point or vertex.

➢ ***To create a plane through a line and point:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.
2. Select an existing axis or edge.
3. Select an existing reference point or vertex. A preview of the new plane is displayed.
4. Select the **Containing Edge**/**Axis** option to create the plane through both the point and the axis/edge.
5. Select the **Normal to Edge**/**Axis** option to create the plane through the point and normal to the axis/edge.
6. Click **OK** to create the plane.

5.1.6 Three Point Plane

You can create a plane through three points or vertices.

➢ ***To create a plane through three points:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.
2. Select an existing point or vertex.
3. Select a second point or vertex.
4. Select the third point or vertex. A preview of the new plane is displayed.
5. Click **OK** to create the plane.

5.1.7 Plane Normal to 3D Sketch or 3D Edge

You can create a plane normal to an open 3D sketch or normal to an edge of a solid model.

➢ ***To create a plane normal to a 3D sketch or edge:***

1. Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog appears.
2. Select an existing open 3D sketch or model edge. A preview of the new plane is displayed.
3. Check the **Other End** box to move the plane to the other end of the sketch or edge.
4. Click **OK** to create the plane.

5.2 Axes

Three axes are visible by default in part and assembly workspaces: the **X-axis**, **Y-axis**, and **Z-axis**. Additional axes can be inserted as needed.

5.2.1 Axis through Axis or Edge

You can create an axis through an existing axis or linear edge.

➢ ***To create an axis through an edge:***

1. Select the **Insert Axis** tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog appears.
2. Select an existing axis or edge. A preview of the new axis is displayed.
3. Click **OK** to create the axis.

5.2.2 Axis through Two Points

You can create an axis through two points or vertices. This method can be used to create an axis at an angle.

➢ ***To create an axis through two points:***

1. Select the **Insert Axis** tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog appears.
2. Select an existing point or vertex.
3. Select the second point or vertex. A preview of the new axis is displayed.
4. Click **OK** to create the axis.

5.2.3 Axis Using Cylindrical Face

You can create an axis using a cylindrical face as reference.

➢ ***To create an axis using a cylindrical face:***

1. Select the **Insert Axis** tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog appears.
2. Select an existing cylindrical face. A preview of the new axis is displayed.
3. Click **OK** to create the axis.

5.2.4 Axis through Two Planes

You can create an axis at the intersection of two existing reference planes or planar faces.

➢ ***To create an axis through two planes:***

1. Select the **Insert Axis** tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog appears.
2. Select an existing reference plane or planar face.
3. Select a reference plane or planar face that intersects the first plane or face. A preview of the new axis is displayed.
4. Click **OK** to create the axis.

5.2.5 Axis Offset and Parallel to Axis or Edge

You can create an axis that is parallel to an existing axis or edge and offset by a specified distance.

➢ ***To create an axis offset and parallel to an edge:***

1. Select the **Insert Axis** tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog appears.
2. Select an existing edge or axis.
3. Select a reference plane or planar face that passes through the edge or axis selected in step 1. A preview of the new axis is displayed.
4. Enter an offset distance.

5. If necessary, select **Reverse** to create the axis in the opposite direction.

6. Click **OK** to create the axis.

5.3 Points

In part and assembly workspaces, the origin is the only 3D point displayed by default. Additional 3D points can be inserted as necessary.

➢ ***To insert a new point:***

1. Select the **Insert Point** tool from the Inspection toolbar; or from the **Insert** menu, select **Point**; or right-click in the work area and select **Insert Point** from the pop-up menu. The **Insert Point** dialog appears.

2. Select the appropriate point type below and follow the steps accordingly.

5.3.1 Point at Specified Coordinates

You can create a point using direct coordinate entry.

➢ ***To create a point using direct coordinate entry:***

1. Enter the 3D coordinates of the new point. The coordinates are based on the absolute coordinate system with the origin located at (0,0,0).

2. Click **OK** to create the point.

5.3.2 Point at Plane and Axis/Edge

You can create a point at the intersection of a plane or planar face and an axis or edge.

➢ ***To create a point at a plane and edge:***

1. Select a reference plane or planar face.

2. Select an axis or edge. A preview of the new point is displayed.

3. Click **OK** to create the point.

5.3.3 Point at Axis/Edge and Axis/Edge

You can create a point at the intersection of two axes or edges.

➢ ***To create a point at the intersection of two edges:***

1. Select an axis or edge.

2. Select a second axis or edge. A preview of the new point is displayed.

3. Click **OK** to create the point.

5.3.4 Point at the Center of Circular Edge

You can create a point at the center of a circular edge.

➢ ***To create a point at the center of a circular edge:***

1. Select a circular edge. A preview of the new point is displayed.

2. Click **OK** to create the point.

5.3.5 Point at Vertex

You can create a point at a vertex.

➢ ***To create a point at a vertex:***

1. Select the vertex. A preview of the new point is displayed.
2. Click **OK** to create the point.

5.3.6 Point along Edge

You can create a point along an edge at a specified location.

➢ ***To create a point along an edge:***

1. Select the edge. A preview of the new point is displayed.
2. Enter a **Ratio** value. A ratio of .5 will place the point at the midpoint of the edge; 1.0 will place the point at the end of the edge.
3. Click **OK** to create the point.

5.3.7 Point between Two Points

You can create a point between two points at a specified location.

➢ ***To create a point between two points:***

1. Select a point or vertex.
2. Select a second point or vertex.
3. Enter a **Ratio** value. A ratio of .5 will place the point at the midpoint between the two existing points or vertices.
4. Click **OK.**

5.4 Reference Surfaces

You can insert surfaces as reference geometry, similar to application planes, in order to trim or extend solids to the surfaces. In addition, you can thicken a reference surface into a solid.

5.4.1 Inserting Reference Surfaces

Surfaces can be inserted from IGES or SAT files only.

➢ ***To insert a surface:***

1. Open the part workspace into which you want to insert the surface.
2. From the **Insert** menu, select **Surfaces**; or select the **Insert Surfaces** tool from the Inspection toolbar. The Insert Surface dialog appears.
3. Select the file that you want to insert and click **Open**. The Insert Surface Options dialog appears, if you have that option turned on.

Note: To turn on the option to see the Insert Surface dialog, from the **Tools** menu, select **Options> Interoperability Tab**. In the Insert Options section, check the box beside **Show Options When Inserting**.

4. Select the desired options and click **OK**.

    - **Stitch Adjoining Faces** takes faces that meet at a common edge and places them in the same surface body. Each resulting lump becomes a surface.
    - **None** inserts the surfaces as they exist in the file. Each lump is a surface. The body is unchanged.
    - **Unstitch to Standalone Surfaces** converts each face into a separate surface.
    - **Heal** is same as healing for import of solid. It will recalculate inaccurate geometry in order to make the part more accurate upon import. It cleans up the body by making sure edges lie on faces, eliminates duplicate vertices, etc. Healing attempts to fix problems detected with the model by changing it. Most IGES files require healing to import properly.
    - **Make Tolerant** will tag inaccurate geometry to enable more intelligent subsequent operations after import. Because the geometry of a tolerant model is allowed to be less precise, inaccurate or leaky data can often be imported using the Make Tolerant option. Making a model tolerant leaves its underlying geometry unchanged.

The surface is inserted into the work area, and is listed in the Design Explorer under both Surfaces and Features.

5.4.2 Positioning Reference Surfaces

The default location of the inserted surface is such that its coordinate system matches the current part workspace coordinate system with the same origin, orientation, and scale. You can modify the location of the surface by positioning it.

➢ ***To position a surface:***

1. In the Design Explorer, right-click the surface you want to position.

2. Select **Edit**. The **Position Surface** dialog appears.

3. In **Method**, select the method you want to use to position the surface.

    - **Axis and Offset:** The surface will be moved the offset distance in the direction of the specified axis or edge.
    - **Planar:** Choose points or vertices to use for the start of the move and the end of the move. These points will be projected onto the chosen plane, and the surface will move parallel to the plane.
    - **Coordinates:** Change the location of the surface by designating X, Y, and Z distances.
    - **Axis and Angle:** Rotate the surface about the specified axis by the given angle.

4. Fill in the appropriate data for the chosen method.

5. Click **OK** to apply the change.

5.4.3 Thickening Reference Surfaces

Once a reference surface has been inserted, you can thicken the surface to transform it into a solid. This will allow geometry such as holes and extruded cuts to be applied to the resulting solid. Only one surface can be thickened at a time.

➢ ***To thicken a surface:***

1. From the **Feature** menu, select **Thicken Surface**; or select the **Thicken Surface** tool from the Part Modeling toolbar. The Thicken Surface dialog appears.

2. In **Surface**, select the surface you want to thicken.

3. In **Thickness**, enter the desired thickness value.

4. In **Direction**, choose forward, reverse, or both sides. Arrows in the workspace will display the direction the thickness will be applied.

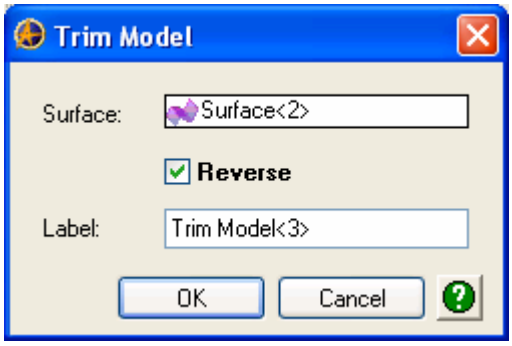
5. Click **OK** to apply the Thicken feature.

5.4.4 Trimming a Solid

You can trim a solid model with respect to a surface.

➢ ***To trim a model:***

1. From the **Feature** menu, select **Trim Model**; or select the **Trim Model** tool from the Part Modeling toolbar. The **Trim Model** dialog appears.



2. Select the **Surface** you want to use to trim the model. Arrows in the workspace will show the direction the surface will trim. Check the **Reverse** box if necessary.

3. Click **OK** to apply the Trim Model feature. All features contained in the model will be trimmed.

5.4.5 Extruding to Geometry

You can use a reference surface in the "To Geometry" option of an extrusion or sweep. See *Feature Creation* (on page 101) for information on extruding "To Geometry".

The sketch you are extruding must lie completely within the outline of the surface or the extrusion will fail.

5.5 Reference Geometry Visibility

You can hide reference geometry on an individual basis, by group, or altogether.

5.5.1 Hiding Individual Reference Geometry Items

➢ ***To hide reference geometry items:***

1. Select the reference geometry item in the Design Explorer or work area. You can also select multiple items. The selected items become highlighted in the Design Explorer as well as the work area.

2. Right-click in the work area and select **Hide** from the pop-up menu; or press **Ctrl + H** on the keyboard; or from the **Edit** menu select **Hide Selection**.

The selected reference geometry items are hidden. The text in the Design Explorer associated with the items will also become light gray.

5.5.2 Hiding Reference Geometry Groups

You can hide groups of planes, axes, points, surfaces, the 3D view indicator, or the world coordinate system independently.

From the **View** menu, select **References > Coordinate System**, **References > 3D View Indicator**, **References > Planes**, **References > Axes**, **References > Points**, or **References > Surfaces**, depending on which group you want to hide. The group becomes hidden, and the text in the Design Explorer associated with the items will also become light gray.

You can also press **Ctrl + Shift + P** on the keyboard to hide planes.

Note: Upon hiding planes as a group, all planes are hidden, except for the plane that is currently selected. To hide all the planes at once, including the selected plane, select the **Planes** node in the Design Explorer, right-click in the work area, and select **Hide** from the pop-up menu.

5.5.3 Hiding All Reference Geometry Groups

You can hide all reference geometry groups at one time.

➢ ***To hide all reference geometry groups:***

From the **View** menu, select **References > All**. All reference geometry groups become hidden, except the plane that is currently selected. The text in the Design Explorer associated with the items will also become light gray.

5.6 Renaming Reference Geometry

New reference geometry items are, by default, sequentially named beginning with 4, e.g. **Plane <4>**, **Axis <4>**, **Point <4>**, etc. You can rename default reference geometry, as well as inserted reference geometry items.

➢ ***To rename reference geometry items:***

1. Right-click the reference geometry item in the Design Explorer and select **Rename** from the pop-up menu. Or, click the reference geometry in the Design Explorer twice, with a short pause between clicks. The name is highlighted and the cursor appears next to the name.

2. Type a new name.

3. Press **Enter** on the keyboard.

5.7 Deleting Reference Geometry

You can delete inserted reference geometry items. You cannot delete default reference geometry.

➢ ***To delete an inserted reference geometry item:***

Right-click the reference geometry item in the Design Explorer and select **Delete** from the pop-up menu; or select the reference geometry item in the Design Explorer or work area and press **Delete** on the keyboard.

5.8 Editing Reference Geometry Properties

You can modify the properties associated with an inserted reference geometry item.

➢ ***To edit an inserted reference geometry item:***

1. Right-click the reference geometry item in the Design Explorer and select **Edit** from the pop-up menu; or right-click the reference geometry item in the work area and select **Edit** from the pop-up menu. The dialog associated with the item appears, displaying the original properties.

2. Modify the properties as necessary.

3. Click **OK** to apply the change.

CHAPTER 6

3D Sketching

3D Sketching allows you to create guide curves for better control of lofts. In addition, using 3D sketches allows you to create sweeps that are ideal for modeling piping and cabling systems.

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6.1 The 3D Sketching Interface

The 3D Sketching toolbar is shown by default on the right side of the workspace. Commonly used sketch tools are accessible on the Sketching toolbar.

Activate 3D Sketch (with options flyout) ... activate 3D sketch mode

Select ... select sketch figures and entities

Constraints (with options flyout) ... place manual constraints on a sketch

Dimension ... place dimensions on sketch figures

Line (with options flyout)... create a line figure

Arc (with options flyout)... create an arc figure

Spline (with options flyout)... create a spline figure

Sketch Node ... create a sketch node

Direct Coordinate Entry ... create sketch figures by entering Cartesian or Polar coordinates

Define Coordinate System ... define the current coordinate system

Cycle Sketch Plane ... change the sketch plane

Elevate ... change the elevation by dragging the cursor

Elevation ... change the elevation by entering coordinates

Fillet ... place a fillet on two existing figures

All of the tools accessible on the 3D Sketching toolbar are also accessible from the **3D Sketch** menu.

6.2 3D Sketching Context

Just as 2D sketching is done in 2D Sketch Mode, 3D sketching takes place 3D Sketch Mode. The 3D sketching environment has a dedicated toolbar and right-click menu, which allow you to access the 3D sketch functions.

You can modify the display of various 3D sketch items from the view menu.

➢ ***To modify the display of sketch items:***

1. From the **View** menu, select **Sketch Display**. The following items can be turned on or off in the display:
    - Grid
    - Sketch Dimensions
    - Constraint Symbols
    - Guide Lines
    - Current Coordinate System Indicator
2. Select an item to turn it on or off. A checkmark next to an item means it is visible (on). This is a toggle on/off.

➢ ***To modify the view orientation:***

From any orientation back to the sketch plane:

From the **View** menu, select **Orient > To Sketch Plane**; or select the **Orient to Sketch Plane** tool from the Orient View Toolbar.

From any orientation to the isometric view of the sketch plane:

From the **View** menu, select **Orient > Isometric To Sketch Plane**; or select the **Isometric To Sketch Plane** tool from the Orient View Toolbar.

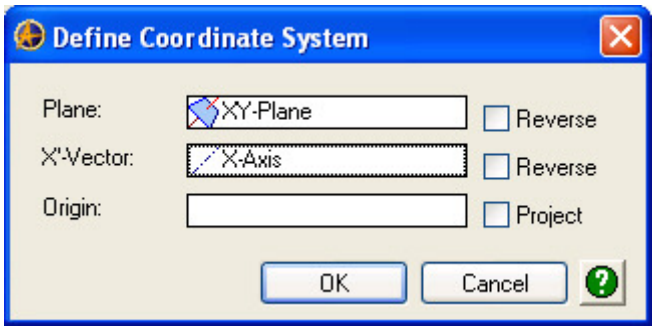
6.3 Current Coordinate System

In the 3D sketching environment, all position-related data is entered with respect to the Current Coordinate System (CCS). The CCS is depicted graphically by a 3D coordinate system in the work area. This graphical coordinate system is called the CCS indicator. Upon entering 3D sketch mode, the CCS is automatically created using the active plane, which becomes the XY-Plane of the CCS. The direction of the axes is determined automatically by the system.

➢ ***To Define A New 3D Coordinate System:***

The CCS can be changed using the Define 3D-Coordinate System command.

1. Select the **Define Coordinate System** tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Define Coordinate System**. The Define Coordinate System dialog appears.



2. Enter the following information:

   - **Plane:** Required - Enter the desired reference plane, planar face, or 2D sketch to be used as the plane. Checking **Reverse** will toggle which **Z-axis** direction is positive.
   - **X-Vector:** Optional - Enter the reference axis, linear edge, 2D sketch line, or 3D sketch line to be used for the X-Vector. Checking **Reverse** will toggle which **X-axis** direction is positive.
   - **Origin:** Optional - Enter a reference point, vertex, 2D node, or 3D node to be used as the origin. If **Project** is checked, the point is projected to the plane to become the origin; otherwise the plane is moved to intersect the point that becomes the origin.

3. Click **OK** to apply the changes.

➢ ***To turn off the CCS indicator:***

From the **View** menu, select **Sketch Display > Current Coordinate System Indicator**. This is an on/off toggle.

6.4 Sketch Plane, Guide Lines, and Elevation

The sketch plane is defined by any of the planes of the CCS and an elevation. The sketch plane can be offset along its normal by an elevation distance.

By default, the sketch plane is the XY-Plane of the CCS. The sketch plane can be altered by using the Cycle Sketch Plane Command.

Coordinate guides are displayed while in 3D sketch to indicate the current location of the cursor with respect to the CCS. Guides for the sketch plane are displayed on the sketch plane. A guide extending from the base plane to the current cursor position indicates the current elevation.

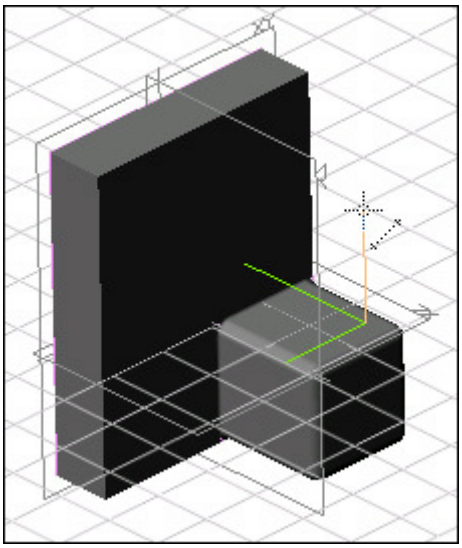


Figure 27: Coordinate Guides Shown with the Sketch Plane Grid

➢ ***To change the sketch plane:***

1. Select the **Cycle Sketch Plane** tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Cycle Sketch Plane**.

2. The sketch plane cycles to the next primary plane each time the tool is clicked. If the grid is turned on, it is displayed on the current base plane.

Notes:

This command can be issued during other commands.

You can also cycle the sketch plane by pressing the Tab or “F” key on the keyboard. If you press one of these keys while in a figure creation command, the cursor will remain in the same position in space, rather than moving to an elevation of zero on the new sketch plane.

Elevation

Elevation is controlled in two ways. The first method is to drag the mouse while in elevate mode. The second method is to use the Elevation Dialog.

➢ ***To control elevation in Elevate mode:***

1. Select the **Elevate** tool from the sketching toolbar; or from the **3D Sketch** menu, select **Elevate**. This puts you into Elevate mode. While in Elevate mode, the coordinates on the base plane remain constant.

2. Click and drag the cursor to the desired height from the base plane.

3. To exit Elevate mode, select the **Elevate** tool again from the sketching toolbar, or from the **3D Sketch** menu, select **Elevate**.

Notes:

To change the elevation of an existing figure, click and drag the figure while in Elevate mode.

Elevate can be issued during other commands.

While in a figure creation command, Elevate can be accessed at anytime by pressing the “E” key on the keyboard and holding it down while placing the figure.

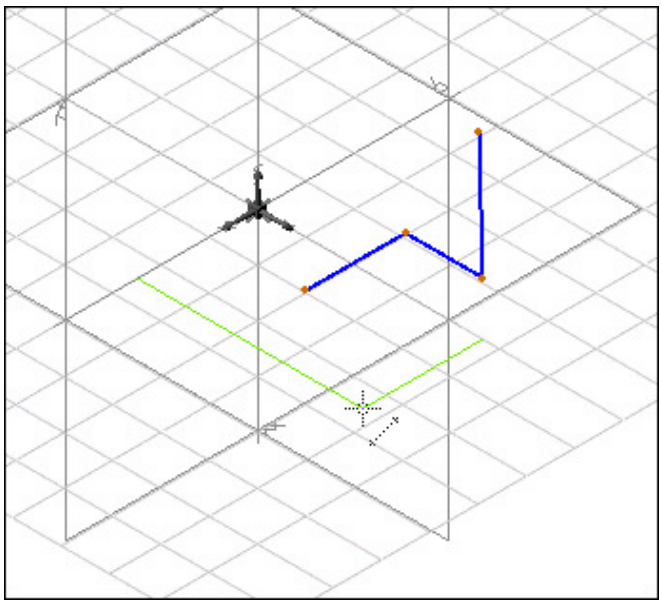


Figure 28: Elevation Height at the Base Plane

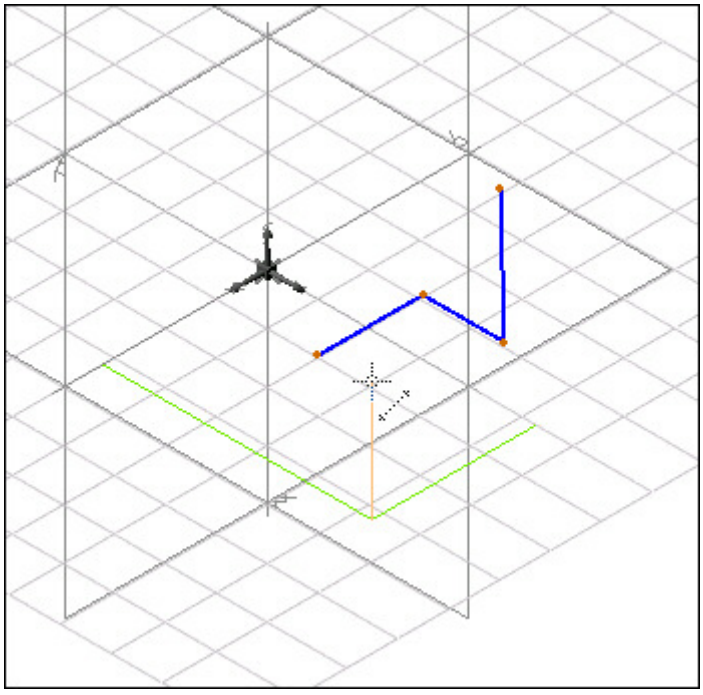


Figure 29: Elevation Height Changed by Dragging Mouse Pointer

➢ ***To control elevation using Elevation Dialog:***

1. Select the **Elevation** tool from the sketching toolbar; or from the **3D Sketch** menu, select **Elevation.** The Elevation dialog appears.

2. Enter the desired elevation in the dialog. This dialog can remain open and in use during other commands.

3. Click the **X** in the upper corner of the dialog to close when finished.

6.5 Entering and Exiting 3D Sketch Mode

6.5.1 Entering 3D Sketch Mode

You must enter 3D sketch mode before you can begin sketching.

➢ ***To enter 3D sketch mode:***

Select the **Activate 3D Sketch** tool from the Sketching toolbar.

Or,

From the **3D Sketch** menu, select **Activate 3D Sketch**.

The activate 3D Sketch tool will appear in the active state while in 3D sketch mode.

6.5.2 Exiting 3D Sketch Mode

The same methods used to enter 3D sketch mode can be used to exit 3D sketch mode.

➢ ***To exit 3D sketch mode, use any one of the following options:***

- Click the **Select** tool from the View toolbar.
- Create a feature from the sketched profile. For example, select a feature tool such as the **Sweep** tool from the Part Modeling toolbar.
- Select the **Regenerate** tool from the Part Modeling toolbar.
- From the **Feature** menu, select **Regenerate All**.

6.6 3D Sketch Figures

6.6.1 Line

➢ ***To sketch a line:***

1. Select the **Line** tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Figures > Line**; or right-click and select **Line** from the pop-up menu.

2. Position the cursor at the location where you want to start the line.

3. Click to start the line and drag the cursor to sketch the line.

4. Click again to complete the line segment. You can continue to sketch additional line segments by clicking. Double-click or press **ESC** on the keyboard to complete the line.

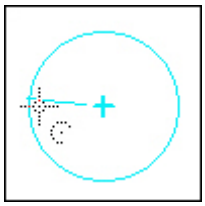
6.6.2 Arc

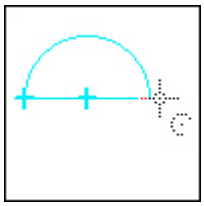
You can sketch three different circular arc types: 1) **Center, Start, End**; 2) **Start, End, Radius**; 3) **Tangent-Start, End**.

➢ ***To sketch a circular arc using Center, Start, End:***

1. Select the **Circular Arc - Center, Start, End** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Center, Start, End**; or right-click and select **Circular Arc** from the pop-up menu.
2. Click in the Work Area to place the center of the arc.
3. Click a second time to start the arc.
4. Move the cursor to sketch the arc.
5. Click a third time to complete the arc.

Note: The plane of the arc is defined by the three nodes. If the three nodes are collinear, the current sketch plane is used.





➢ ***To sketch a circular arc using Start, End, Radius:***

1. Select the **Circular Arc - Start, End, Radius** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Start, End, Radius**.
2. Position the cursor at the arc starting location.
3. Click to start the arc.
4. Click a second time to locate the end of the arc.
5. Move the cursor to size the arc.
6. Click a third time to complete the arc.

Note: The plane of the arc is defined by the three nodes.

➢ ***To sketch a circular arc using Tangent-Start, End:***

1. Select the **Circular Arc - Tangent-Start, End** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Tangent-Start, End**.
2. Click a line or circular arc.
3. Move the cursor to size the arc.
4. Click a second time to complete the arc.

Note: The plane of the arc is defined by the tangent line passing through the nodes at the start and end of the arc.

You can also modify a 3D sketch circular arc in the same way as a *2D sketch circular arc* (see "Circular Arcs" on page 31).

6.6.3 Spline

Creation of NURBS curves by interpolation

Using this method, specify a set of nodes in the Work Area to define the spline. A curve is then interpolated based on the placement of the nodes.

1. Select the **Spline** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Spline.**
2. Click in the Work Area to start the spline curve.
3. Move the cursor and click a second time to place an interpolation node.
4. Move the cursor to shape the curve.
5. Continue clicking to place additional nodes and curve segments.

   Note: One or more interpolation nodes may be specified via the direct coordinate entry tool.

6. Double-click or hit escape to complete the spline curve.

   Note: If you place the final node at the same location as the first, the spline will be completed as a closed spline.

➢ ***To edit a Spline Curve:***

In 3D Sketch, you can edit spline curves by selecting and dragging any of the nodes in the curve.

➢ ***Inserting a Node Into a Spline Curve***

1. From the **3D Sketch** menu, select **Insert Node into Spline,** or select the **Insert Node into Spline** tool from the sketching toolbar.
2. Click the spline to place a node.
3. Continue clicking the spline to place as many nodes as desired.
4. Choose the **Select** tool to exit the Insert Node command.

6.7 3D Sketch Nodes

6.7.1 Placing a Sketch Node

➢ ***To place a sketch node:***

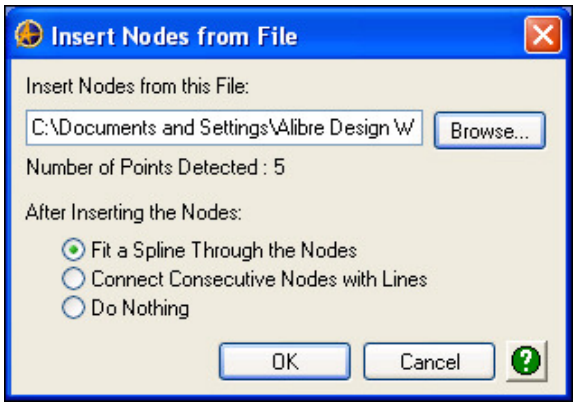
1. Select the **Sketch Node** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Node**.
2. Click in the work area to place a sketch node. You can continue clicking to place as many sketch nodes as you need.

6.7.2 Inserting Sketch Nodes from a File

You can insert sketch nodes from a comma-delimited text file. These nodes can then be used to create features such as Sweeps.

➢ ***To insert sketch nodes from a file:***

1. From the **3D Sketch menu**, select **Figures > Insert from File**. The **Insert Nodes from File** dialog appears.



2. Enter the required information:

   File Name: Type in the file name or use the Browse button to designate the file containing the nodes.

   Choose one of the following:

   - Fit a Spline Through the Nodes - the system will interpolate a spline through the nodes.
   - Connect Consecutive Nodes with Lines - each node will be connected by a straight line.
   - Do Nothing - the nodes will be placed in the sketch without connecting figures.

3. Click **OK** to insert the nodes.

6.8 Working with Existing 3D Sketch Figures

6.8.1 Adding Fillets

➢ ***To add a fillet to a sketch figure:***

1. Select the **Fillet** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Fillet**. The **Fillet Figures** dialog appears.

2. Select the first figure by clicking it. The figure name appears in the **Figures to Fillet** field in the dialog.

3. Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** field in the dialog.

4. Enter the fillet radius value in the **Radius** field.

5. Click **Apply** to create the fillet. The Fillet Figures dialog remains open so you can continue to place fillets on additional figures.

   Note: Consecutive fillets with a diameter equal to the first fillet will not be dimensioned; instead an equal constraint will automatically be placed.

6. Click **Close** to close the Fillet Figures dialog.

6.8.2 Dimensioning 3D Sketch Figures

To fully define the 3D sketch you must place dimensions on sketch figures. 3D sketch dimensions function similar to 2D sketch dimensions. It is not required to dimension sketches before they are used to create features. In addition, sketch dimensions can easily be changed and modified at any time.

Linear, radial, and angular dimensions can be created. Linear dimensions are placed on a plane running through an axis created by the endpoints and parallel with the screen at the time of creation. The dimension will always lie on this plane from that time forward. The dimension can be dragged along this plane from place to place. Radial dimensions may be created for circular arcs. Angular dimensions may be created between two non-parallel lines.

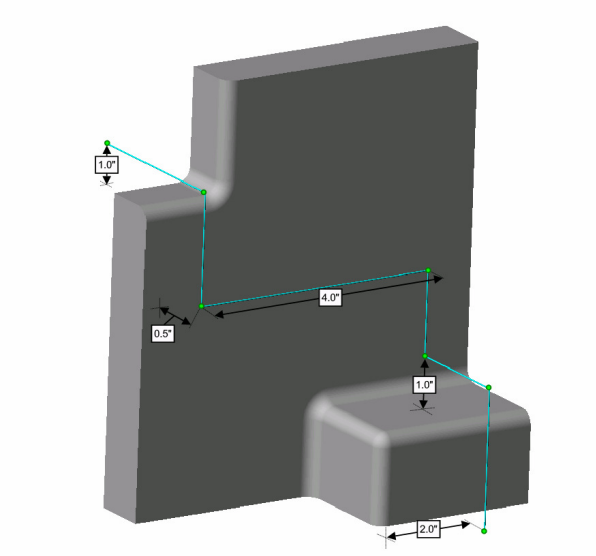


Figure 30: Examples of 3D Linear Dimensions

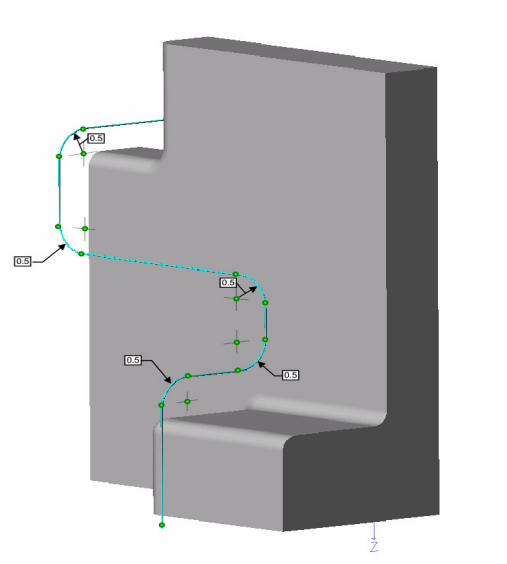


Figure 31: Examples of 3D Radius Dimensions

➢ ***To dimension individual figures:***

Follow the steps below to place length dimensions on lines, and diameter dimensions on arcs and circles.

1. Select the **Dimension** tool from the Sketching toolbar; or from the **Sketch** menu, select **Dimension**; or right-click in the work area and select **Dimension** from the pop-up menu.

2. Move the cursor over the figure you want to dimension. The figure is highlighted.

3. Click the figure to show a preview of the dimension. Move the cursor to move the dimension preview.

Note: You can press the **Esc** key on the keyboard to cancel the current dimension operation.

4. After the dimension has been positioned properly, click again. A dimension control box appears.

5. Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.

Note: You can enter fractions (e.g. **3**/**8**) and simple equations (e.g. **1.5 * 3**) into the dimension control box. You can also enter a value with units other than the current display units (e.g. **5 mm**). The value will be converted to the display units automatically. Supported unit abbreviations are **"**, **'**, **mm**, **cm**, and **m**.

➢ ***To dimension distances or angles between figures:***

Follow the steps below to place distance dimensions or angular dimensions between figures, e.g. between lines, between nodes, between arc or circle center nodes, etc.

1. Select the **Dimension** tool from the Sketching toolbar, or from the **Sketch** menu, select **Dimension**, or right-click in the work area and select **Dimension** from the pop-up menu.

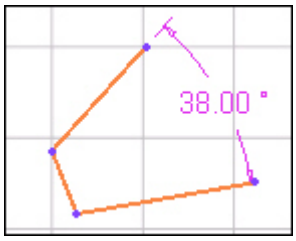
2. Move the cursor over the first figure from which you want to dimension.

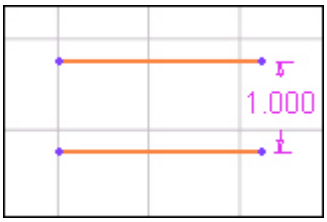
3. Click the figure. A preview may appear but do not place the dimension at this time.

4. Move the cursor over the second figure you want to dimension to and click again. A new dimension preview appears.

5. Move the cursor to position the dimension and click a third time. A dimension control box appears.

6. Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.





6.9 3D Sketch Constraints

Figures in a 3D sketch may be constrained to a size, orientation, and relationship to another figure or model edge. These constraints are similar to their 2D sketch constraint counterparts.

Tip: When you are active in 3D sketch mode, you can constrain sketch figures to other existing sketches. This is particularly helpful when you are sketching guide curves for lofts, to ensure that your guide curves touch each profile used in the loft.

6.9.1 Inferred Constraints

Location

During the creation of a figure, a Location constraint will be applied automatically when a 3D node is created while hovering over the following objects:

- Reference Points
- Vertices
- An Existing 3D Node
- An Edge
- 2D Nodes (in visible 2D sketches that have not been used to create 3D geometry)
- 2D Figures (in visible 2D Sketch that have not been used to create 3D geometry)

This constraint is associative. If the object associated with the constraint moves, the corresponding 3D node moves with it. Inferred constraints can be broken by holding the CTRL key, then clicking and dragging the node away.

Fixed Direction

During the creation of lines that are horizontal or vertical with respect to the sketch plane, an implied fixed direction constraint will be placed on them.

6.9.2 Explicit Constraints

Explicit constraints are applied in a fashion analogous to their 2D equivalents through the use of a toolbar button or menu selection. The Constraints fly-out toolbar is available from the Sketching toolbar.

Coincident - A node can be constrained so that it lies on a figure, model edge, or planar face.

Fixed - Figures may be constrained to a fixed position in the sketch. After the constraint is applied, the node or figure may not be moved without first deleting the constraint.

Direction - The direction of a 3D line can be constrained so that it is held constant.

Parallel - Lines can be constrained to be parallel to each other. In addition, a line can be constrained to be parallel to a linear edge, a reference plane, or a planar face. For planes and faces, the constraint indicates that the line is perpendicular to the normal of the plane or face.

Perpendicular - Lines can be constrained to be perpendicular to other lines, reference planes, reference axes or planar faces. Splines can be constrained to reference planes and planar faces.

Note: With the Perpendicular constraint, the behavior of lines and splines differs: Lines will be parallel to the normal of a plane or face. Splines will be set tangent at the start or end of the spline parallel to the normal of a plane or face.

Tangent - Figures can be constrained to be tangent to a curve. Tangent constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Tangent Continuous - An open 3D figure can be constrained to be continuous at its endpoint to another open 3D figure or an open edge.

Collinear - Figures may be constrained so that they lie in the same line.

6.10 Other 3D Sketch Functions

Other sketch options are similar to 2D sketch options:

The sketch grid (on page 66)

Snapping to the working plane (on page 67)

Cursor display (on page 68)

Editing sketches (on page 73)

Renaming sketches (on page 73)

CHAPTER 7

Feature Creation

Parts are modeled by creating features. Features are individual 3D shapes representing common mechanical design elements, like bosses and holes, which either create material or remove material in a part. Many features, such as extrude boss and revolve boss, require an associated sketch to define the 2D profile of the 3D shape. Other features, such as fillet and edge chamfer, can be created without a sketch and are applied to existing edges and faces.

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7.1 The Part Modeling Interface

The Part Modeling toolbar is shown by default on the right side of the workspace. Commonly used modeling tools are accessible on the Part Modeling toolbar.

Extrude Boss ... create an extrude boss feature, which adds material to the model in a linear fashion

Extrude Cut ... create an extrude cut feature, which removes material in a linear fashion

Revolve Boss ... create a revolve boss feature, which adds material in a circular fashion

Revolve Cut ... create a revolve cut feature, which removes material in a circular fashion

Loft Boss ... create a loft boss feature, which adds material between 2 or more cross sections

Loft Cut ... create a loft cut feature, which removes material between 2 or more cross sections

Sweep Boss ... create a sweep boss feature, which adds material as a profile is swept along a path

Sweep Cut ... create a sweep cut feature, which removes material as a profile is swept along a path

Helical Boss ... create a helical boss feature, which adds material in a helical fashion

Helical Cut ... create a helical cut feature, which removes material in a helical fashion

Fillet (with options fly-out) ... create a fillet feature (fly-out includes **Edge Chamfer ...** create an edge chamfer feature)

Push/Pull Face/Sketch (with options fly-out) ... modify features by dragging faces and/or edges of a sketch (fly-out includes additional Push/Pull tools as well as **Remove Tool** ... modify features by removing faces; and **Draft Surface** ... add a draft to a face on the model

Hole ... create a hole feature

External Cosmetic Thread ... Apply an external cosmetic thread to a cylindrical face. This thread information can be displayed in drawings.

Shell ... create a shell feature

Linear Feature Pattern (with options fly-out) ... create a pattern of a feature in one or two linear directions (fly-out includes **Circular Pattern**, as well as **Linear and Circular Topology Patterns** ... create a pattern of a portion of the model)

Feature Mirror ... mirror a feature about an edge or axis

Scale ... scale a part model in any direction

Design Boolean (with options flyout)... create Boolean feature

Trim Model (with options fly-out) ... trim a model with respect to a reference surface (fly-out includes **Thicken Surface** ... thicken a reference surface)

Insert Catalog Feature ... insert a feature that you previously saved

Equation Editor ... open the Equation Editor

Regenerate ... regenerate the part to update changes

The tools that are accessible on the Part Modeling toolbar are accessible from the **Feature** menu as well. The Feature menu also contains tools that do not have a corresponding toolbar icon.

- **Thin Wall Boss > Extrude** . . . create a thin wall extrude boss feature, which adds material to the model in a linear fashion
- **Thin Wall Boss > Revolve** . . . create a thin wall revolve boss feature, which adds material in a circular fashion
- **Thin Wall Boss > Sweep** . . . create a thin wall sweep boss feature, which adds material as a profile is swept along a path
- **Thin Wall Cut > Extrude** . . . create a thin wall extrude cut feature, which removes material in a linear fashion
- **Thin Wall Cut > Revolve** . . . create a thin wall revolve cut feature, which removes material in a circular fashion
- **Thin Wall Cut > Sweep** . . . create a thin wall sweep cut feature, which removes material as a profile is swept along a path
- **Chamfer > Vertex** . . . create a vertex chamfer feature
- **Offset Face** . . . offset a face by a specified distance
- **Move Face** . . . move a face by a specified distance
- **Save Catalog Feature** ... save a feature for use in other models

7.2 Feature Terminology

7.2.1 Feature Types

Boss

Boss features are used to create or add material in a part. Generally, the first feature you create in a part will be a boss feature.

Cut

Cut features are used to remove material from a part. You cannot create a successful cut feature until at least one boss feature has been created.

Note: Most 3D features will be created from a sketch. You can only use an individual sketch to create one feature. Once a sketch is used to create a feature, it will appear under that feature in the Design Explorer. You cannot use it again, unless you delete the 3D feature that is using it. In the figure below, Sketch <1> is used by Revolution<1>, Sketch<2> is used by Extrusion<2>, and Sketch<8> is unused. Therefore, only Sketch<8> is currently available for use in a new feature.

7.3 Extrude Boss and Extrude Cut

Although the extrude boss and extrude cut features are different in end result, the steps used to create them are identical.

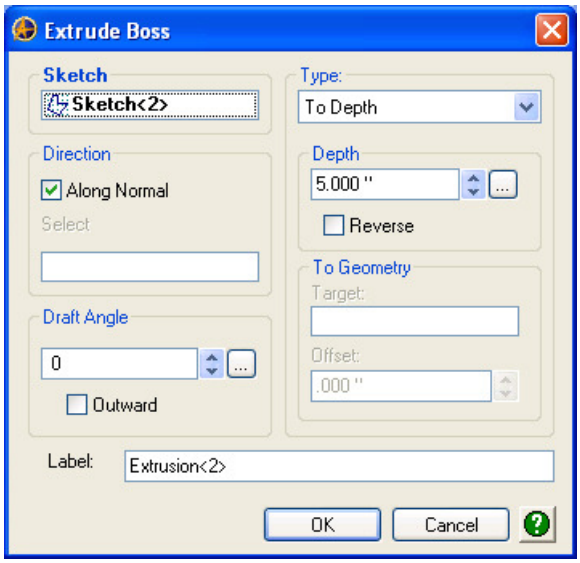
Extrude features, also referred to as **extrusions**, either create or remove material by extending a sketch in a linear direction by a specified distance.

7.3.1 Creating Extrude Boss and Extrude Cut Features

Extrude features require a *closed sketch* (see "Open and Closed Sketches" on page 69).

➢ ***To create an extrude boss or extrude cut:***

1. With a sketch still active, select the **Extrude Boss** tool or the **Extrude Cut** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Extrude** or **Cut > Extrude**. The **Extrude Boss** or the **Extrude Cut** dialog appears.



2. Click in the **Sketch** field to activate it, then the sketch you want to extrude.

3. Select a **Type** from the pull down menu. Follow the steps below depending on the Type.

   To Depth, Mid Plane, Through All extrusions

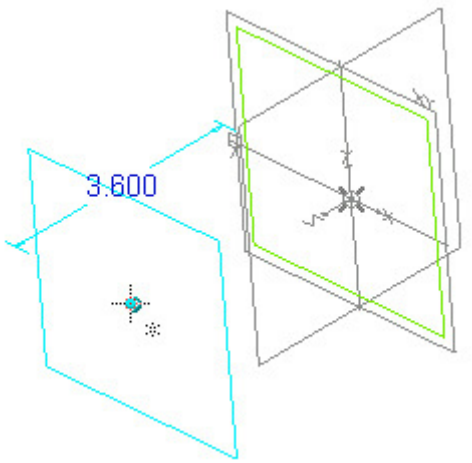
   To Depth - Creates an extrusion of a specified depth on one side of the sketch plane.

   Mid Plane - Creates an extrusion of a specified depth on both sides of the sketch plane. Half the extrusion length is proportional to each side of the sketch plane.

   Through All - Applies only to extrude cut. Creates the cut through the entire solid in the specified direction.

   a. Specify the extrusion **Depth** value. If using **To Depth**, you can select the **Reverse** option if necessary to create the extrusion on the opposite side of the sketch plane.

Note: When using the **To Depth** or **Mid Plane** options, you can dynamically resize the extrusion in the work area by dragging the node associated with the sketch profile. As you drag the node, the extrusion length will automatically increase or decrease increments based on the **Spinner Increment** value (**File** > **Properties** > **Units** tab).



b. To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the extrusion direction.

c. Specify a **Draft Angle** if required, and click the **Outward** option to change the direction of the draft, if necessary.

d. You can enter a custom **Label** if you would like to customize the feature name as it is displayed in the Design Explorer.

e. Click **OK** to create the extrusion.

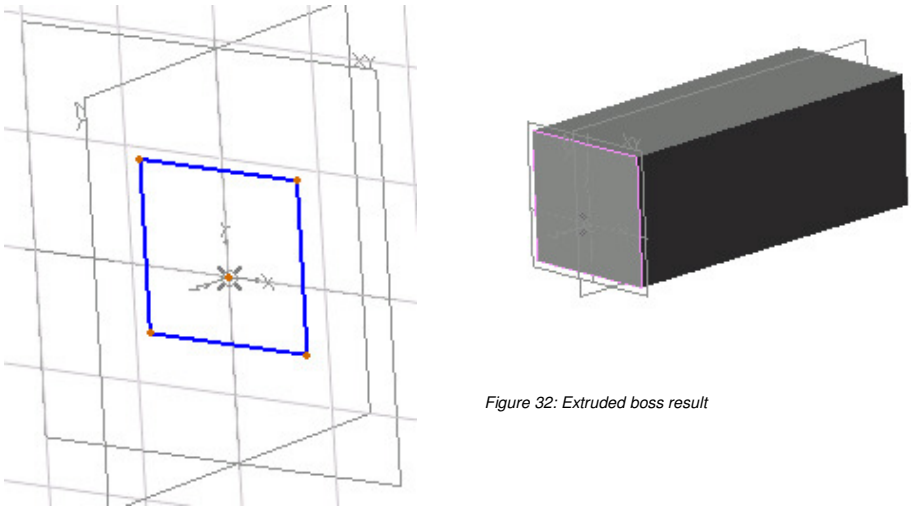


Figure 32: Extruded boss result

To Geometry extrusions

To Geometry - Creates an extrusion up to another reference plane or face.

f. Select a **Target** by selecting a reference plane, surface, or face.

g. You can also specify an **Offset** value to create the extrusion up to a specified distance from the Target.

h. To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the new extrusion direction.

Specify a **Draft Angle** if required, and click the **Outward** option to change the direction of the draft, if necessary.

i. You can enter a custom **Label** if you would like to customize the feature name as it is displayed in the Design Explorer.

j. Click **OK** to create the extrusion.

To Next extrusions

To Next - Creates an extrusion up to the nearest face(s) of the part. For a To Next Extrusion to be successful, the sketch must lie within the boundaries of the Next Face. If it is not, an error message will be displayed and the feature will fail.

a. Select the **Reverse** option, if necessary, to create the extrusion on the opposite side of the sketch plane.

b. To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the new extrusion direction.

c. Specify a **Draft Angle** if required, and click the **Outward** option to change the direction of the draft, if necessary.

d. You can enter a custom **Label** if you would like to customize the feature name as it is displayed in the Design Explorer.

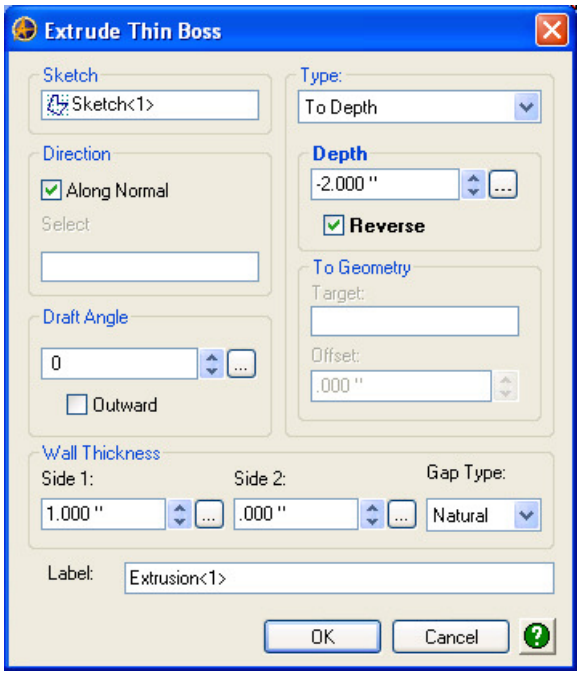
e. Click **OK** to create the extrusion.

7.3.2 Creating Thin Wall Extrude Boss and Cut Features

Thin wall extrude features either create thin walled bosses or cuts by extending a sketch in a linear direction by a specified distance. As opposed to normal extrude boss and cut features discussed previously, thin wall extrusions can be created with *open or closed sketches* (see "Open and Closed Sketches" on page 69).

➢ ***To create a thin wall extrude boss or cut:***

1. With a sketch still active, from the **Feature** menu, select **Thin Wall Boss > Extrude** or **Thin Wall Cut > Extrude**. The **Extrude Thin Boss** or the **Extrude Thin Cut** dialog appears.



2. If the **Sketch** field is not already populated with the correct sketch, click in the Sketch field to activate it, then select the sketch you want to use to create the feature.

3. Select a **Type**. (Refer to *Creating Extrude Boss and Cut* (see "Creating Extrude Boss and Extrude Cut Features" on page 103) for information related to the Type condition.)

4. Specify a **Length/Depth** value or select a **Target**.

5. Select a **Gap Type**:

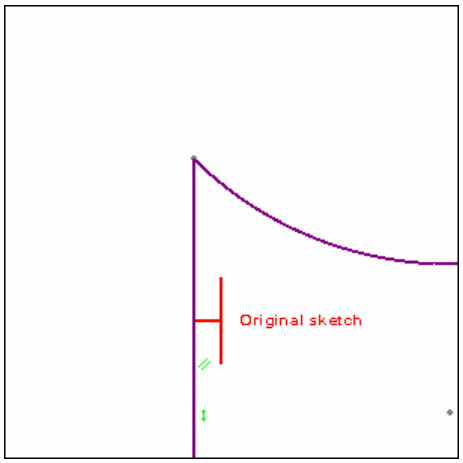


Figure 33: Original Sketch Example

Natural: Extends the edges of the wall along their natural curves until they intersect.

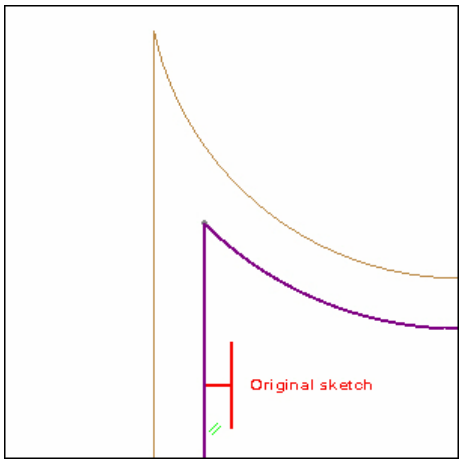


Figure 34: Result Using Natural Gap Type

Round: Creates fillets on any corners of the wall profile.

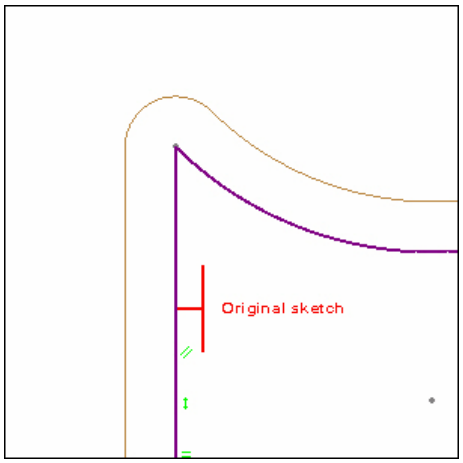


Figure 35: Result Using Round Gap Type

Extend: Extends the edges of the wall beyond their endpoints in straight lines until they intersect.

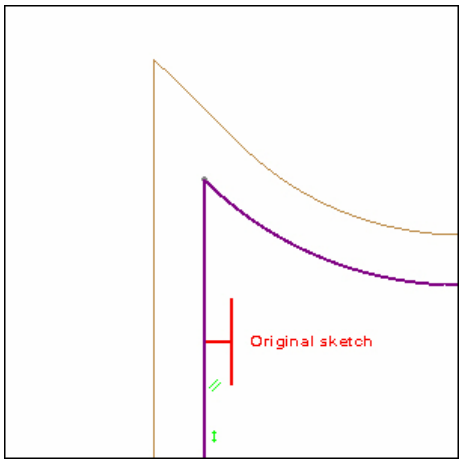


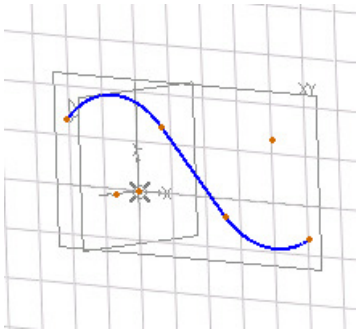
Figure 36: Result Using Extend Gap Type

6. Specify an extrusion direction. Select the **Along Normal** option to create the extrusion normal to the sketch plane. Deselect the **Along Normal** option and select an edge or axis to create the extrusion in a direction other than normal to the sketch plane.

7. To taper the extrusion, specify a **Draft Angle**. To change the draft orientation, select the **Outward** option.

8. Specify the **Wall Thickness** by entering values for the **Side 1** and **Side 2** wall thicknesses. Specifying a **Side 1** value will create or remove material inward from the sketch, and a **Side 2** value will create or remove material outward from the sketch.

9. Click **OK** to create the thin wall extrusion.



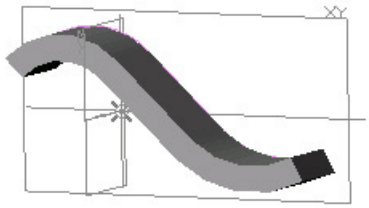


Figure 37: Resulting thin wall boss extrusion

7.4 Revolve Boss and Revolve Cut

Although the revolve boss and revolve cut features are different in end result, the steps used to create them are identical.

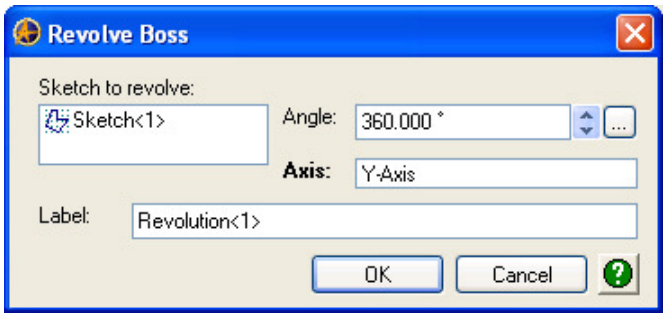
Revolve features, also referred to as **revolutions**, either create or remove material by revolving a sketch by a specified angle around a centerline.

7.4.1 Revolve Boss and Revolve Cut Features

Revolve boss and cut features require a *closed sketch* (see "Open and Closed Sketches" on page 69).

➢ ***To create a revolve boss or cut:***

1. With a sketch still active, select the **Revolve Boss** or the **Revolve Cut** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Revolve** or **Cut > Revolve**. The **Revolve Boss** or the **Revolve Cut** dialog appears.



2. If the **Sketch to revolve** field is not populated with the sketch you want to revolve, click in it to activate it, then select the sketch you want to revolve.

3. Select an edge, axis, or sketch line as the **Axis** centerline.

4. Specify the rotation **Angle**.

5. Enter a custom **Label** to modify how the feature name is displayed in the Design Explorer.

6. Click **OK** to create the revolution.

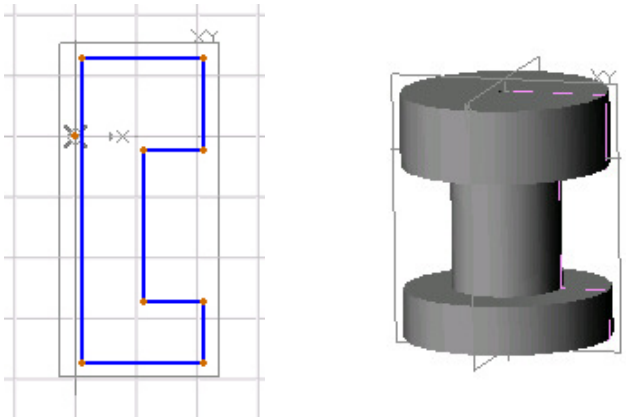


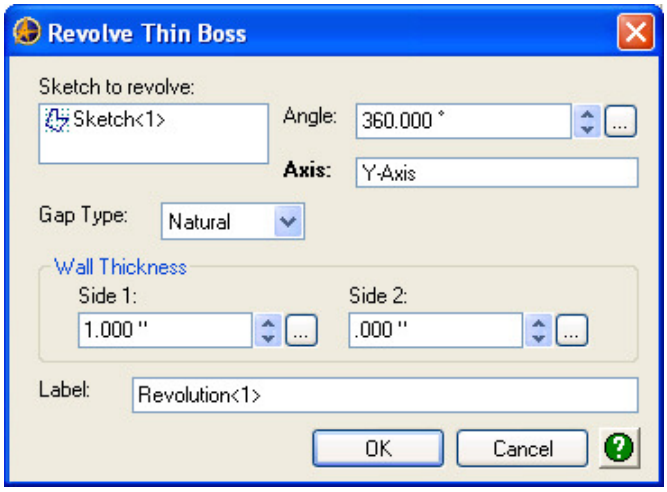
Figure 38: Resulting revolve boss feature

7.4.2 Thin Wall Revolve Boss and Cut Features

Thin wall revolve features either create or remove a thin wall of material by revolving an open or closed sketch around a centerline.

➢ ***To create a thin wall revolve boss or cut:***

1. With a sketch still active, from the **Feature** menu, select **Thin Wall Boss > Revolve** or **Thin Wall Cut > Revolve**. The **Revolve Thin Boss** or the **Revolve Thin Cut** dialog appears.



2. Specify the rotation **Angle**.

3. Select an edge, axis, or sketch line as the **Axis** centerline.

4. Choose a **Gap Type**:

   Natural: Extends the edges of the wall along their natural curves until they intersect.

   Round: Creates fillets on any corners of the wall profile.

   Extend: Extends the edges of the wall beyond their endpoints in straight lines until they intersect.

5. Specify the **Wall Thickness**. **Side 1** creates or removes material on the inward side of the sketch; **Side 2** creates or removes material on the outward side of the sketch.

6. Click **OK** to create the thin wall revolution.

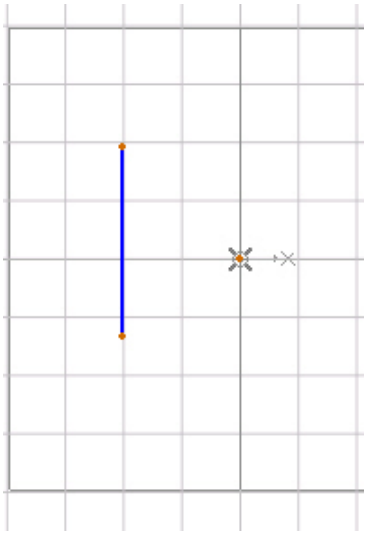


Figure 39: Initial sketch for a revolved thin wall boss

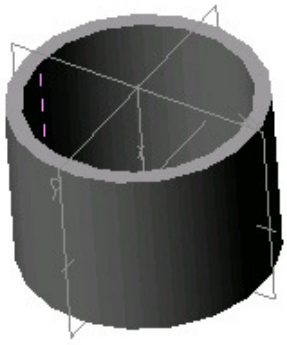


Figure 40: Resulting revolved thin wall boss

7.5 Loft Boss and Loft Cut

Although the loft boss and loft cut features are different in end result, the steps used to create them are identical.

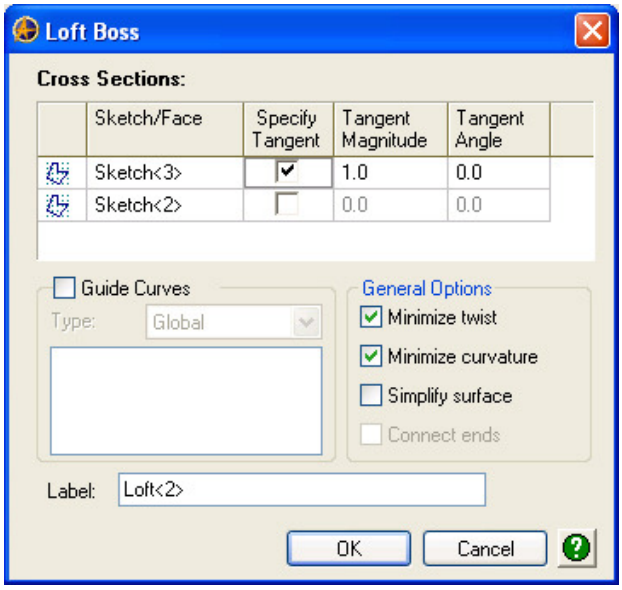
Loft features, simply referred to as lofts, either create or remove material by forming a feature that transitions from one cross section (such as a sketch or existing face) to another along a path. The two sketches reside on different planes, but do not have to be parallel to each other.

You can use two or more sketches or existing faces to create lofts.

➢ ***To create a loft boss or cut:***

1. Sketch at least two profiles on two different planes. The sketches must be closed. The planes do not have to be parallel. You may loft to a point by creating a sketch with exactly one node in it.

2. Select the **Loft Boss** tool or **Loft Cut** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Loft** or **Cut > Loft**. The **Loft Boss** or the **Loft Cut** dialog appears.



3. In the work area, select the sketches or faces to use in the loft. If you select inside the design explorer, you must press **Shift** to select multiple sketches or faces. As you make the selections, the corresponding labels appear in the **Cross Sections** box.

   The sketches/faces must be listed in the order in which the loft will be created. To remove a sketch or face from the box, select it from the list and press **Delete** on the keyboard.

4. If desired, check the **Specify Tangent** box next to a sketch or face. When “Specify Tangent” is selected, local control of surface directions in the vicinity of the lofted sketches/faces is possible. If “Tangency Angle” is 0 ˚, surface normals remain perpendicular to the sketch plane normal, or parallel to the adjoining face normals.

5. If Specify Tangent is checked, Tangent Angles can be specified for sketches (but not faces). Specify the desired angle in the **Tangent Angle** field. In this case, the lofted surface tangent would be at the angle specified with the sketch plane normal.

6. If Specify Tangent is checked, additional control is obtained using the Tangent Magnitude for each Cross Section. Tangent Magnitudes control the rate at which surfaces diverge from the cross section sketch planes or faces. As Tangent Magnitudes increase, surfaces diverge more slowly from surface tangents at the sketch cross sections. Specify the desired value in the **Tangent Magnitude** field. A value of 0 is equivalent to not using “Specify Tangent”.

7. If desired, Guide curves can be specified to constrain the location or direction of the lofted surface that is generated. To specify Guide curves, check the **Guide Curves** checkbox, and select the sketches in the work area. 2D and 3D sketches may be used (but you cannot use guide curves if you are using a face as one of your cross sections).

   Tip: When you are active in 3D sketch mode, you can constrain sketch figures to other existing sketches. This is particularly helpful when you are sketching guide curves for lofts, to ensure that your guide curves touch each profile used in the loft.

   Guide curves are each exactly one open or closed loop, each touching all the profiles. Multiple guide curves can be used for Global and Local types, however, for the Tangent type, only one guide curve should be specified. As each curve is selected, the corresponding labels appear in the Guide Curves list box. To remove a Guide curve from the box, select it from the list and press the **Delete** key on the keyboard, or choose an option from the right-click menu.

   Guide curve types:

   Global: Creates “virtual” guide curves, providing the ability to make one guide curve globally affect the lofted surfaces.

   Local: These guide curves provide the ability to make one guide curve locally affect the lofted surfaces.

   Tangent: Forces the loft surface to follow the tangent vector of the guide curve at the point where the curve intersects the profile's plane.

8. If necessary, select a loft creation option:

   - **Simplify Surface:** Converts the resultant loft face from a spline type to an analytic type when possible. A simplified surface contains less data, is of higher quality, and is faster to process in subsequent modeling operations.
   - **Minimize twist:** Aligns the profiles so that the start of the second sketch is aligned to the start of the first sketch.
   - **Connect ends:** The first cross-section is treated as if it is also the last cross-section. This is not available when using Guide Curves, as the Guide Curves can be used to connect the first and last cross sections. You must have at least three cross-sections to use this option.
   - **Minimize Curvature:** This determines the Tangent Magnitudes based on maximizing the minimum radius of curvature of the lofted body as a whole. This not only helps to create more pleasing surfaces, but also ensures greater ability to shell and blend lofted models. This option is applicable only when a single guide curve is present and the Guide Curve Type is "tangent."

9. Click **OK** to create the loft.

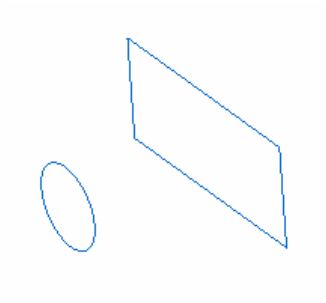


Figure 41: Cross Sections for Loft

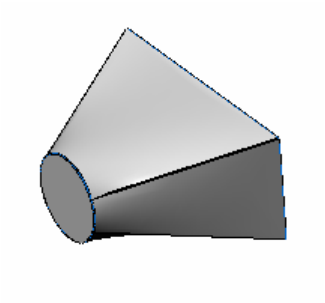


Figure 42: Resulting Loft Boss Feature

The following pictures illustrate the results of a loft created using the same sketches, but with different types of guide curves.

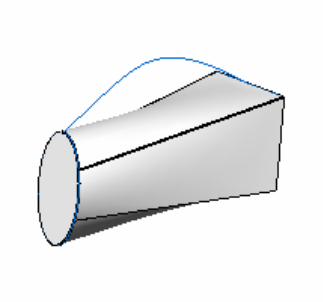


Figure 43: Loft Using No Guide Curve

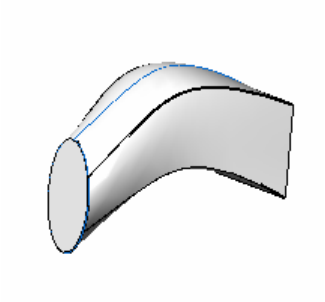


Figure 44: Loft with Global Guide Curve

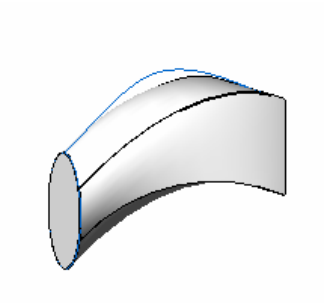


Figure 45: Loft with Tangent Guide Curve

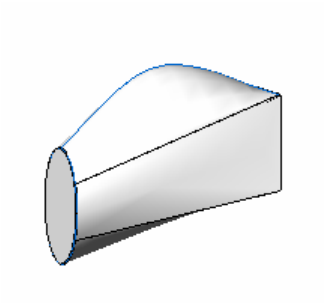


Figure 46: Loft with Local Guide Curve

7.6 Sweep Boss and Sweep Cut

Although the sweep boss and sweep cut features are different in end result, the steps used to create them are identical.

Sweep features, simply referred to as sweeps, either create or remove material by moving a sketch along a path defined by a second sketch.

The following guidelines should be followed when creating sweep features:

- The sketch that defines the profile must be *closed* (see "Open and Closed Sketches" on page 69).
- The sketch(es) that define the path can be open or closed but cannot be self-intersecting.

- The sketch path cannot lie on the same sketch plane as the profile.
- The sketch path must either start on the profile plane or pass through the profile plane.
- The sketch path can be multiple 2D or 3D sketches, as well as Edges of Parts or Surfaces.
- The sketch path must be continuous.

7.6.1 Sweep Boss and Sweep Cut Features

➢ ***To create a sweep boss or cut:***

1. Sketch the closed profile.

2. Sketch the path(s). The path must either start on or pass through the plane of the profile, but is not required to pass through the profile itself. The path cannot intersect itself.

3. Select the **Sweep Boss** tool or the **Sweep Cut** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss** > **Sweep** or **Cut** > **Sweep**. The **Sweep Boss** or the **Sweep Cut** dialog appears.

4. In the **Sketch to Sweep** field, select the profile sketch.

5. In the **Path Objects** field, select the path sketch(es). One or more sketches or edges may be selected.

6. Select a **Sweep Type**. You can create the sweep along the **Entire Path** or **To Geometry**.

7. If **To Geometry** was selected, click in the **Geometry Target** box, and select an existing plane, surface, or face in the work area. You can also specify a **Geometry Offset** value to create a gap between the target and the end of the feature.

8. If **Entire Path** was selected, click the **Rigid** checkbox, if desired, to force the profile to remain parallel to the profile's sketch plane through the sweep.

9. To place a draft on the sweep, specify a **Draft Angle**. Select the **Outward** option if necessary. You cannot create a draft if the **Rigid** option is applied.

10. Click **OK** to create the sweep.

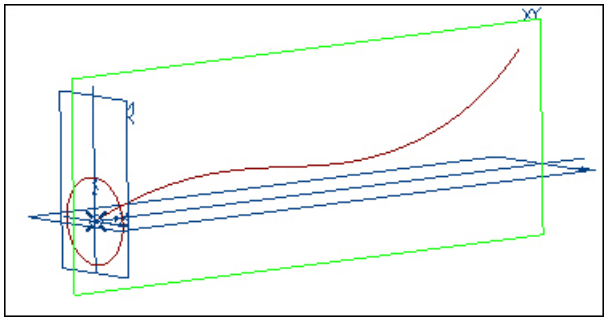


Figure 47: Sweep Profile and Path Sketches

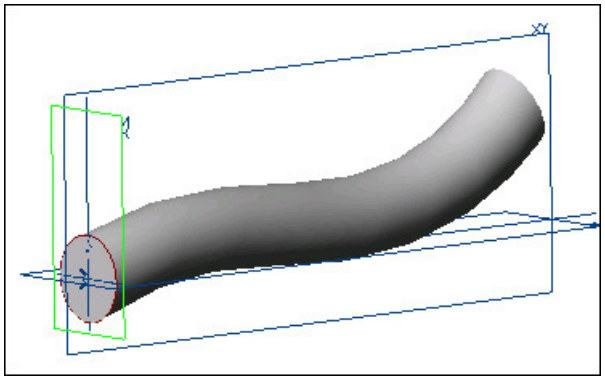


Figure 48: Resulting Sweep

7.6.2 Thin Wall Boss Sweep and Cut Sweep Features

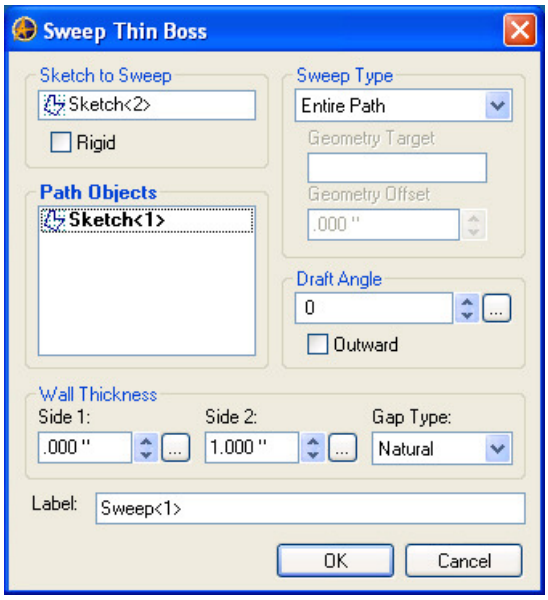
Thin wall sweep features either create or remove a thin wall of material by moving a closed or open sketch along a path defined by a second sketch.

➢ ***To create a thin wall boss or cut sweep:***

1. Sketch a closed or open profile to define the sweep cross-section.

2. On a different plane, create a new sketch and sketch the path.

3. From the **Feature** menu, select **Thin Wall Boss > Sweep** or **Thin Wall Cut > Sweep**. The **Sweep Thin Boss** or the **Sweep Thin Cut** dialog appears.



4. Select the **Sketch to sweep** and the **Path Objects** (these are the sketch or sketches you want to use for the path).

5. Select a **Sweep Type**. You can create the sweep along the **Entire Path** or **To Geometry**.

6. If **To Geometry** was selected, click in the **Geometry Target** box, and select an existing plane, surface, or face in the work area. You can also specify a **Geometry Offset** value to create a gap between the target and the end of the feature.

7. If **Entire Path** was selected, click the **Rigid** checkbox, if desired, to force the profile to remain parallel to the profile's sketch plane through the sweep.

8. Select a **Gap Type**:

   Natural: Extends the edges of the sweep wall along their natural curves until they intersect.

   Round: Creates fillets on any corners of the wall profile.

   Extend: Extends the edges of the wall beyond their endpoints in straight lines until they intersect.

9. Specify the **Wall Thickness**. **Side 1** creates or removes material on the inward side of the sketch. **Side 2** creates or removes material on the outward side of the sketch.

10. To taper the extrusion, specify a **Draft Angle**. To change the draft orientation, select the **Outward** option.

11. Click **OK** to create the thin wall sweep.

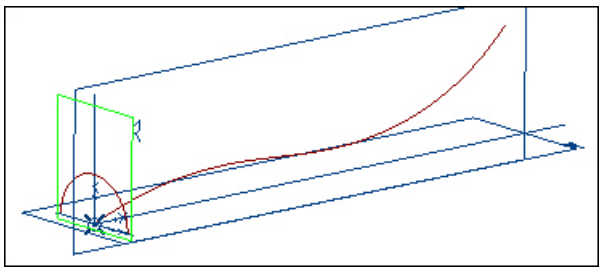


Figure 49: Thin Wall Boss Sweep Profile and Path Sketches

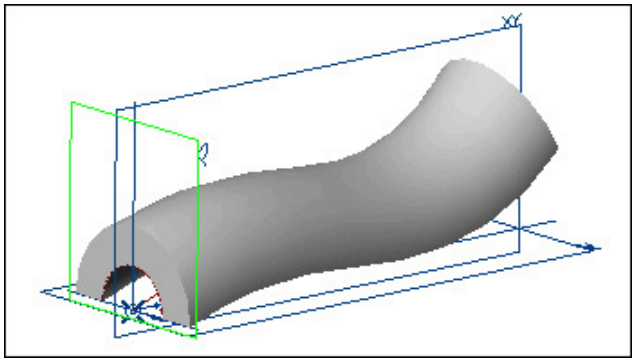


Figure 50: Resulting Thin Wall Sweep Boss

7.7 Helical Boss and Helical Cut

Although the helical boss and helical cut features are different in end result, the steps used to create them are identical.

Helical features, often referred to as helixes, either create or remove material by automatically sweeping a cross section, represented by a sketch, along a helical path. The helical path is automatically created by the software and is driven by user specified parameters. Helical features are beneficial when modeling springs, internal and external threads, ball screws, worm gears, etc.

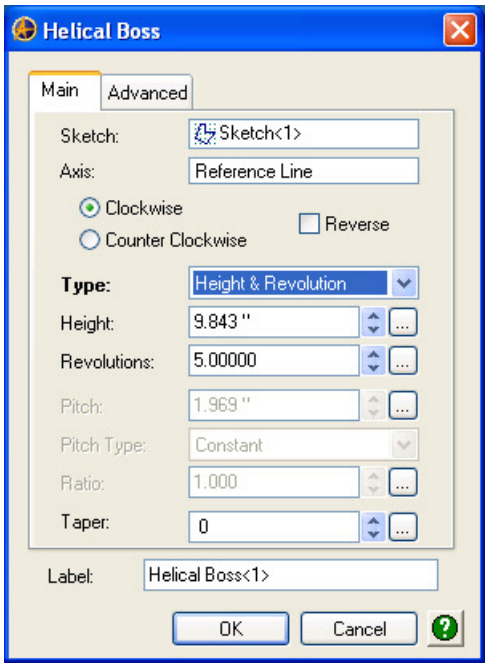
The following guidelines should be followed when creating helical features:

- The sketch that defines the cross-section of the helix must be *closed* (see "Open and Closed Sketches" on page 69).
- The sketch that defines the cross-section of the helix must also contain a **reference line** that represents the helical axis.

➢ ***To create a helical boss or cut:***

1. Sketch the cross-section of the helix. The cross-section is not limited to a certain profile but must be a closed sketch.
2. From the **Sketching** toolbar, select the **Reference Line** tool.
3. Sketch a reference line of any length in the axial direction in which the helix will be created. This reference line will represent the axis of the helix.
4. If necessary, place a dimension between the reference line and sketch figure(s) created in step 1.

5. Select the **Helical Boss** or the **Helical Cut** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Helix** or **Cut > Helix**. The **Helical Boss** or the **Helical Cut** dialog appears, and the **Main** tab is initially displayed by default.



6. In the **Sketch to Revolve** field, select the sketch representing the helical cross-section. The **Reference Line as Axis** field is automatically populated with the reference line located within the sketch. If more than one reference line exists, you can choose which one you want to use.

7. Select the direction of the helix, **Clockwise** or **Counter Clockwise.**

8. If necessary, click the **Reverse** checkbox to change the axial direction of the helix.

9. From the **Type** pull-down menu, select the appropriate helix type:

    - **Height and Revolution**: a helix is generated by specifying the overall feature height, as well as the number of helical revolutions within the specified height.
    - **Height and Pitch**: a helix is generated by specifying the overall feature height, as well as pitch. The **pitch** is defined as the distance from one point on the helix to a corresponding point on the next revolution measured parallel to the axis.
    - **Revolution and Pitch**: a helix is generated by specifying the number of revolutions, as well the pitch.
    - **Spiral**: a flat helix is generated by specifying the number of revolutions, as well as pitch.

10. Specify the appropriate helix parameters (**height, pitch, revolutions)** depending on which type was selected. Note that if **Pitch** is specified, you can also specify **Constant**, **Variable Ratio**, and **Variable End** pitch conditions.

    - **Constant**: Lets you maintain a constant distance. The "ratio" control is disabled.
    - **Variable End**: Specifies the pitch at the end of the helix.
    - **Variable Ratio**: Allows you to change the pitch from start to finish in a ratio such that pitch at the end will be: ratio x start pitch.

11. If applicable, specify a **Taper** angle to create a tapered helix.

12. If applicable, click the **Advanced** tab to specify start and end conditions other than the **Natural** default condition.

    - **Natural** or **Flat** Conditions: for each of the two ends of the coil. The ends can have different end conditions. If Flat is chosen, you will also need to specify the **Transition Angle** and the **Flat Angle**.
    - **Transition Angle**: The distance (in degrees) over which the coil achieves the transition (normally less than one revolution). The example shows the top with a natural end and the bottom end with a one-quarter turn transition (90 degrees) and no flat angle.

    - **Flat Angle**: The distance (in degrees) the coil extends after transition with no pitch (flat). Provides transition from the end of the revolved coil to a flattened end. The example shows the same coil as the Transition angle shown above, but with a half-turn (180 degree) flat angle specified.

13. Specify a **Parallel** or **Normal Profile Orientation** condition.

14. Click **OK** to create the helical feature.

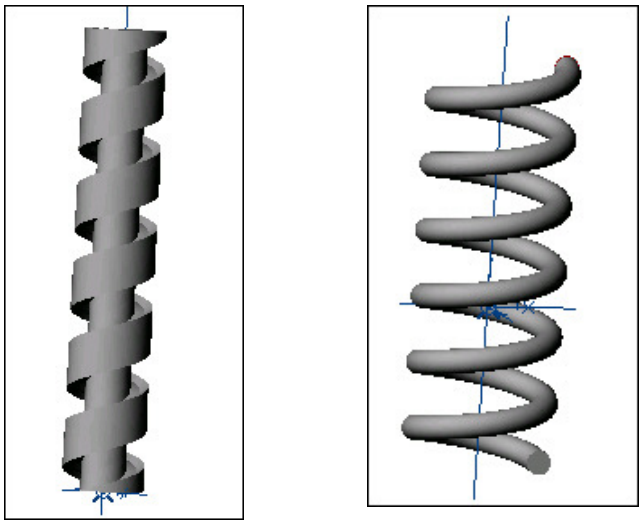


Figure 51: Helical Cut Feature

7.8 Fillet

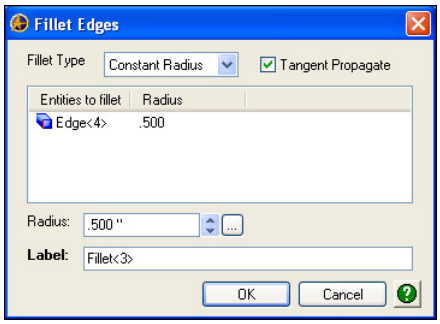
Fillet features create a rounded face on a part. You can place a fillet on individual edges and all edges of a face. You can create a constant radius fillet or a variable radius fillet.

7.8.1 Constant Radius Fillets

Constant radius fillets create a rounded face of constant radius on an edge.

➢ ***To create a constant radius fillet:***

1. Select the **Fillet** tool from the Part Modeling toolbar; or from the **Feature** menu select **Fillet**; or right-click and select **Add Fillet** from the pop-up menu. The **Fillet Edges** dialog appears.



2. Select **Constant Radius** as the **Fillet Type**.

3. Select the edge(s) or face(s) to be rounded. Selecting a face will subsequently select all the edges associated with that face.

4. Deselect the **Tangent Propagate** option, if necessary. The Tangent Propagate option creates a fillet on the selected edge, as well as any other edges that form a path in which a tangent condition can be resolved.

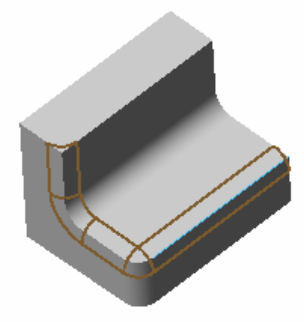


Figure 52: Tangent Propagate On

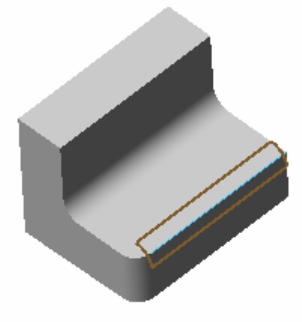


Figure 53: Tangent Propagate Off

5. Specify the **Radius**.

6. Click **OK** to create the fillet.

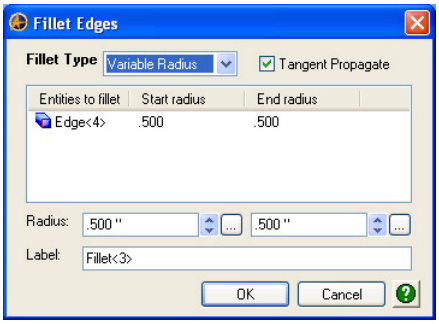
7.8.2 Variable Radius Fillets

Variable radius fillets create a rounded face or variable radius on an edge. You specify a start radius and end radius and a smooth transition is made between the two along the selected edge(s).

➢ ***To create a variable radius fillet:***

1. Select the **Fillet** tool from the Part Modeling toolbar; or from the **Feature** menu select **Fillet**; or right-click and select **Add Fillet** from the pop-up menu. The **Fillet Edges** dialog appears.

2. Select **Variable Radius** as the **Fillet Type**.



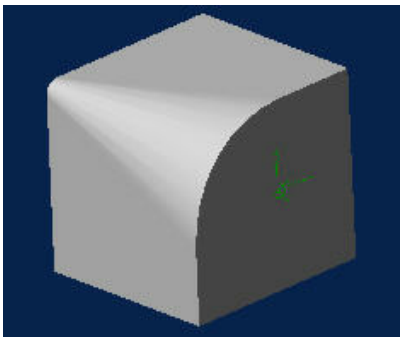
3. Select the edge(s) or face(s) to be rounded. Selecting a face will subsequently select all the edges associated with that face.

4. Deselect the **Tangent Propagate** option if necessary. The Tangent Propagate option creates a fillet on the selected edge, as well as any other edges that form a path in which a tangent condition can be resolved.

5. Specify the start **Radius** in the first radius field.

6. Specify the end **Radius** in the second radius field.

7. Click **OK** to create the fillet.



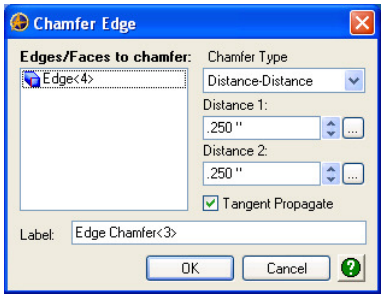
7.9 Chamfers

Chamfer features create a beveled face on a selected edge, face, or vertex.

7.9.1 Edge Chamfers

➢ ***To create an edge chamfer:***

1. Select the **Chamfer** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Chamfer > Edge**; or right-click and select **Add Edge Chamfer** from the pop-up menu. The **Chamfer Edge** dialog appears.



2. Select the edges or faces to chamfer.

3. Select the **Chamfer Type** from the list:

Distance - Distance: specify two distances, one for either side of the chamfer edge.

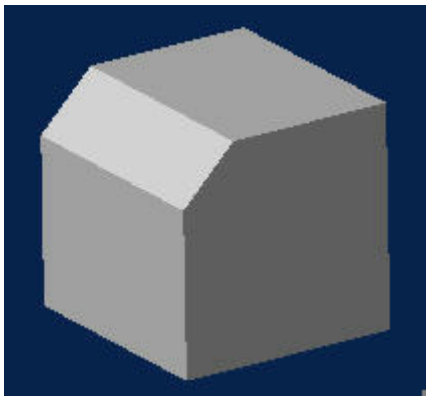
Angle - Distance: specify a distance and angle for the chamfer.

Equal Distance: specify an equal distance for either side of the chamfer edge.

4. Specify the chamfer **Distance(s)** and/or **Angle** values accordingly.

5. Deselect the **Tangent Propagate** option if necessary. The Tangent Propagate option creates a chamfer on the selected edge as well as any other edges that form a path in which a tangent condition can be resolved.

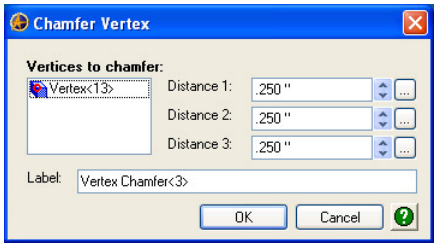
6. Click **OK** to create the chamfer.



7.9.2 Vertex Chamfers

➢ ***To create a vertex chamfer:***

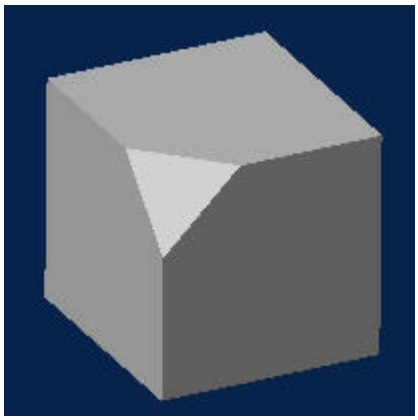
1. From the **Feature** menu, select **Chamfer > Vertex**. The **Chamfer Vertex** dialog appears.



2. Select the vertex or vertices to chamfer.

3. Specify the three **Distance** values for the three chamfer edges.

4. Click **OK** to create the chamfer.

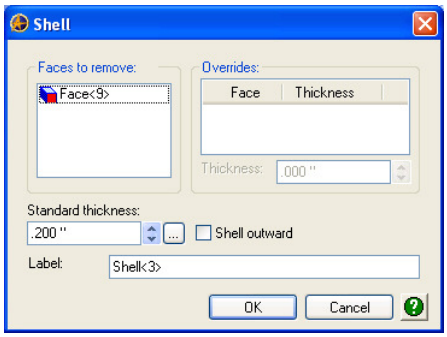


7.10 Shells

Shell features hollow out a part, removing the face(s) you select and leaving thin walls on the remaining faces. The shell can be simple (a single, uniform wall thickness) or complex (multiple, non-uniform wall thickness).

➢ ***To create a shell:***

1. Select the **Shell** tool from the Part Modeling toolbar; or from the **Feature** menu select **Shell**. The **Shell** dialog appears.



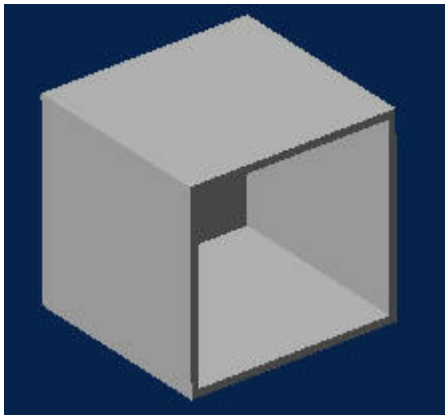
2. Select the **Faces to remove**. You can select multiple faces if necessary.

3. Specify the **Standard thickness** value, which defines the wall thickness after the part is shelled.

4. If required, you can also select faces to override, in which case you can specify a custom wall **Thickness**. To do so, click in the **Overrides** area, and select the face or faces to override in the Design Explorer or work area. You can then set a custom override **Thickness** for each applicable face.

5. Select the **Shell outward** option if you want to add external wall thickness.

6. Click **OK** to create the shell.

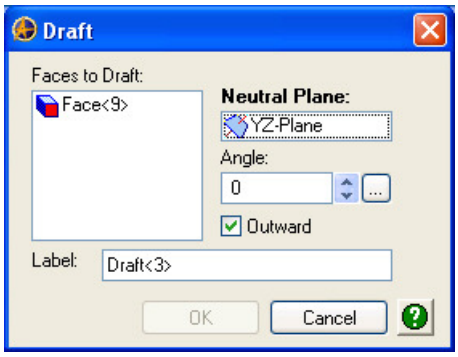


7.11 Draft Faces

Draft surface features create a tapered face at a specified angle to another face in the model. You can create drafts as individual features or you can specify drafts while creating other features, such as extrude bosses.

➢ ***To create a draft:***

1. Select the **Draft Surface** tool from the Part Modeling toolbar; or from the **Feature** menu select **Draft**; or right-click and select **Add Draft** from the pop-up menu. The **Draft** dialog appears.



2. Select the **Faces to Draft**.
3. Select the Neutral Plane. A reference plane or an existing face can be used as the Neutral Plane.
4. Specify the draft Angle.
5. If necessary, check the Outward option to create the draft in the opposite direction.
6. Click OK to create the draft.

7.12 External Cosmetic Threads

Note: Modeling external threads using the Helical Feature tools for items such as screws is not the preferred workflow. This is because helical geometry is very expensive in terms of system resources; when many items, such as fasteners, have helical geometry, it can dramatically slow down your modeling experience.

Typically you will not need to actually see the screw threads - most often they simply need to be accurately represented in a 2D drawing, but not necessarily in the 3D workspace. To accomplish this, you can create External Cosmetic Threads. External Cosmetic Threads are features you can put on cylindrical faces and effectively show that a thread has been defined, without the overhead of modeling the threads. You can also enter callout information. Both the external thread and the callout information can be displayed in a 2D drawing.

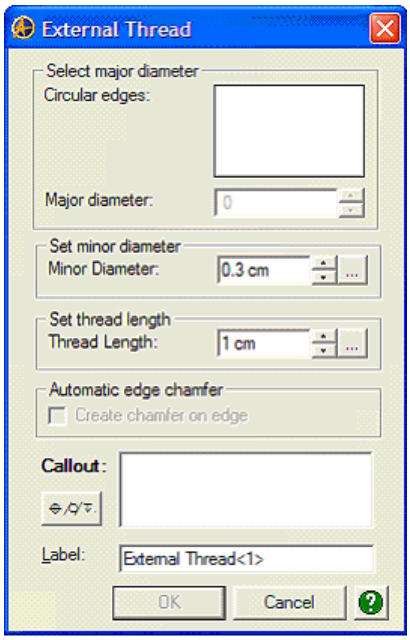
➢ ***To create an external thread:***

1. Click **Feature** on the menu bar and select **External Cosmetic Thread**.

   Or

   Select the item from the toolbar:

2. The External Cosmetic Thread dialog appears.



3. Select the circular edges on the model. This typically represents the end of a screw or other threaded item. You can select multiple edges, however, they must all be exactly the same diameter.

Note: The **Major Diameter** is read-only and represents the diameter of the edge you selected.

4. Set the **Minor Diameter** of the thread.

5. Set the **Thread Length** of the thread.

6. Select **Create Chamfer on Edge** if you need the 2D representation of the thread to have a chamfer on the end.

7. Enter the **Callout** you would like to appear on the drawing.

8. Rename the Feature by changing the **Label**, if desired.

9. Click **OK**.

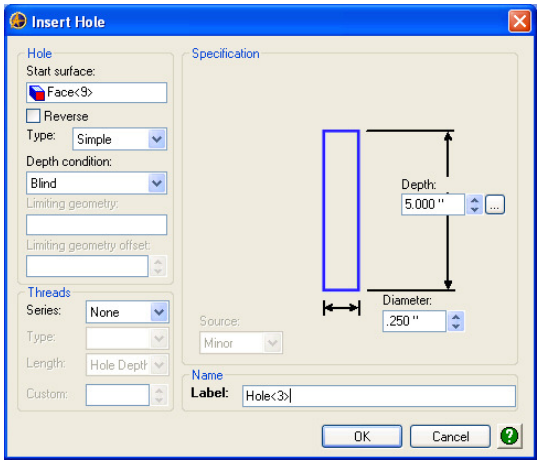
You will notice that as you change the values, the 3D model will update with a general preview of the thread's dimensions.

7.13 Holes

Hole features create standard holes, e.g. counter bored or counter sunk holes, in a part. You place hole features on planar faces, and then place dimensions or constraints on the hole center to position the hole as required. In addition, you can add thread information, which can be shown on drawings.

➢ ***To create a hole:***

1. Select the **Hole** tool from the Part Modeling toolbar; or from the **Feature** menu select **Hole**; or right-click and select **Add Hole** from the pop-up menu. The **Hole** dialog appears.



2. Select the **Start surface**. A reference plane or planar face can be used as the start surface.

Note: You can place multiple holes simultaneously. With the Hole dialog open, click again on the start surface to place another hole.

3. Select the hole **Type** from the list. A number of standard hole types are available.

4. In the **Specification** field, enter the parameters associated with the hole type.

5. Select a **Depth condition** from the list:

   Blind: creates the hole to a specified depth.

   To Limit Geometry: creates a hole up to a specified face or plane. An offset can also be specified.

   Through All: creates a hole through the entire part.

6. Specify the depth parameters, depending on which depth condition is being used.

7. If needed, choose the **Thread** type, as described below.

8. Click **OK** to create the hole(s).

➢ ***To add threads to a hole:***

1. In the **Insert Hole** dialog, choose the desired thread **Series**. Alibre Design supports UNC, UNF, UNEF, UNS, Metric Course, Metric Fine, Metric Special, and NPT series threads.

   Note: Alibre Design does not include pre-defined thread definitions for all of these thread types, although they are supported. You can create your own thread definitions by editing the Alibre Design thread definition file, alibre_unicode.thd. You can use **Notepad** to edit this file. A definition of the file format is embedded within the file. This file is located in the folder **C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files**.

2. Choose the **Type** thread desired.

3. Specify the **Length** of the thread. By default, the thread will extend the entire length of the hole.

4. Choose the **Source** to specify the diameter to use for modeling the hole. You may choose from the minor, pitch, major, and drill tap diameters.

5. Click **OK** to create the hole and define the thread.

Note: The thread information is now included with the hole data. While a graphical representation of the threads is not displayed on the model, when the 2D detailed drawing is created from the 3D design, the threaded hole information can be automatically called out in the applicable orthographic view. See *Displaying Hole Callouts and Threads in Views* (see "Threads and Callouts" on page 271).

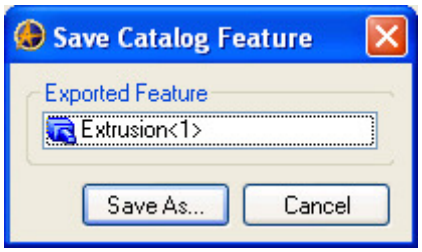
7.14 Catalog Features

Extrude boss and cut features, hole features, sheet metal cut and dimple features, and sketches can be cataloged and reused in the same and other designs.

7.14.1 Saving Catalog Features

➢ ***To save a catalog feature:***

1. From the **Feature** menu, select **Save Catalog Feature**. The Save Catalog Feature dialog appears.



2. Select the feature or sketch to save **from the Design Explorer**.

3. Click **Save As**. The **Save As** dialog appears.

4. Use the **Document Browser** embedded in the **Save As** dialog to navigate to the desired location in the file system. When you click **Save**, the feature will be saved to this location.

5. In **Name**, enter a name to give this feature.

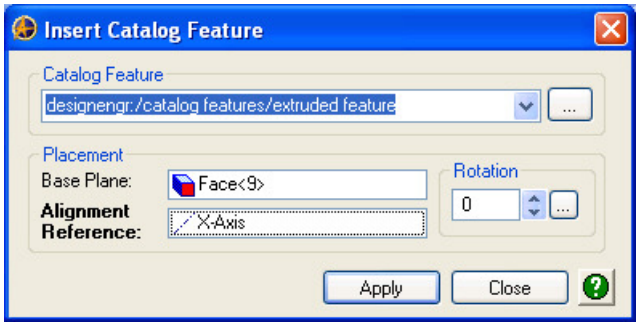
6. Click **Save** to complete the save.

Once you have a saved feature or sketch, you can create several instances of this feature in any model by using **Insert Catalog Feature**.

7.14.2 Inserting Catalog Features

➢ ***To insert a catalog feature after it has been saved:***

1. From the **Feature** menu, select **Insert Catalog Feature**. The Insert Catalog Feature dialog appears.



2. Type in the saved name of the feature or sketch you want to use, or click the '…' button to bring up the 'Insert Design' dialog.

3. In the **Insert Design** dialog, select the feature or sketch to use and click **OK**.

4. In **Placement**, select the 'Base Plane' field in the dialog. Select the face on which you want to insert the feature or sketch.

5. In **Alignment Reference,** select a linear edge to orient the catalog feature. How this feature is placed will depend on how it was created. When exported, the alignment reference is the bottom of the window of the sketch. Therefore, the "bottom" of the feature as it was originally sketched will be aligned with the edge selected here.

6. Enter a value for rotation if you need to rotate the imported operation. You can also use the 'Rotate' and 'Move' handles in the work area to locate the feature - two dots appear on the preview. Move your cursor over them to get the move or rotate arrows. When you get the arrows, click and hold to drag the feature around

7. Click **Apply** and **Close**. You will see in the new feature or sketch in the Feature Tree. The new operation and the sketch created by the catalog feature is a first class operation that can be edited/deleted.

7.15 Copying Existing Features

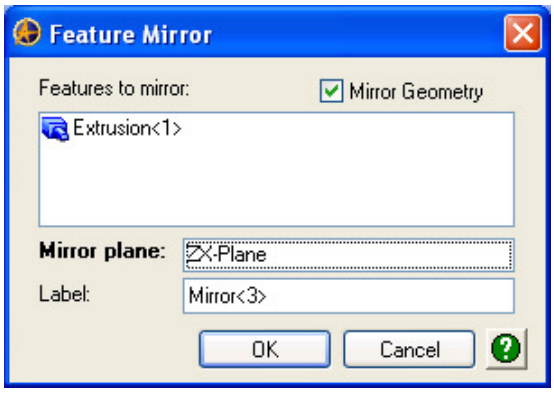
After features have been created, they can be mirrored or patterned to easily create copies.

7.15.1 Mirroring Features

Mirroring a feature creates a copy of a feature mirrored about a reference plane or planar face. If the original feature changes, the mirrored copy will automatically change as well.

➢ ***To mirror a feature:***

1. From the **Feature** menu select **Mirror**; or, select the **Feature Mirror** tool from the Part Modeling toolbar. The **Feature Mirror** dialog appears.



2. Select the **Features to mirror** from the work area or the Design Explorer.

3. Select the **Mirror plane**. You can use a reference plane or planar face.

4. Check the **Mirror Geometry** box if you want to mirror the geometry exactly. You will normally be concerned with this option when you are mirroring a feature that has been created using references to existing geometry such as “extrude to geometry." Below are two models. The first shows a mirror created with the Mirror Geometry box unchecked, and the second with the box checked.

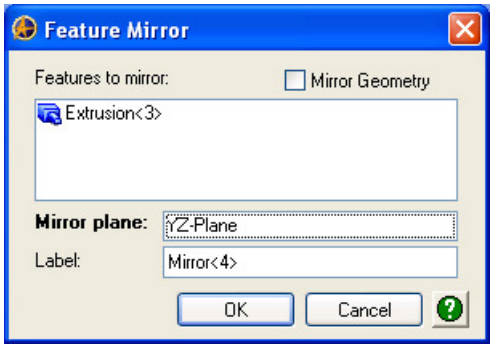


Figure 54: Feature Mirror Dialog Showing Mirror Geometry Unchecked

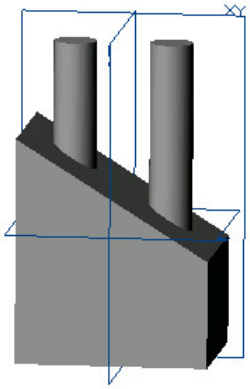


Figure 55: Example of Feature Mirrorred Without Mirror Geometry Option

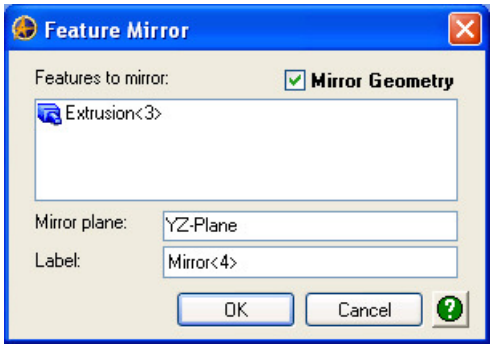


Figure 56: Feature Mirror Dialog Showing Mirror Geometry Checked

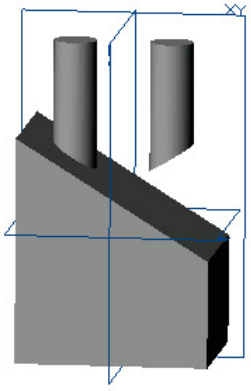


Figure 57: Example of Feature Mirrorred Using Mirror Geometry Option

5. Click **OK** to create the mirror.

7.15.2 Feature Patterns

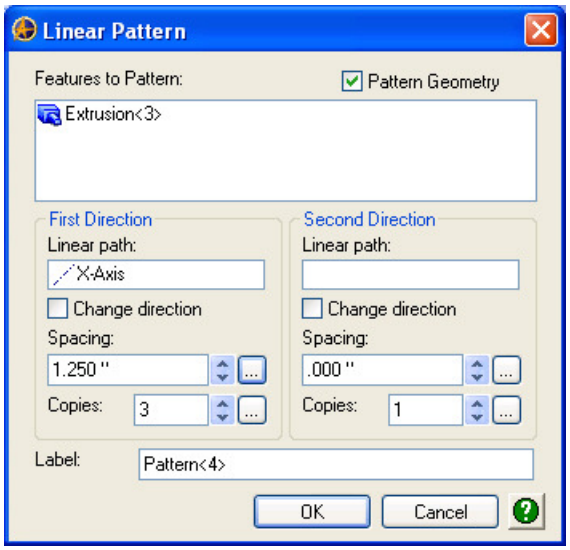
Feature patterns create copies of a feature by repeating it in a linear or circular array.

Linear Patterns

You can use a linear pattern to repeat a feature in one or two linear directions.

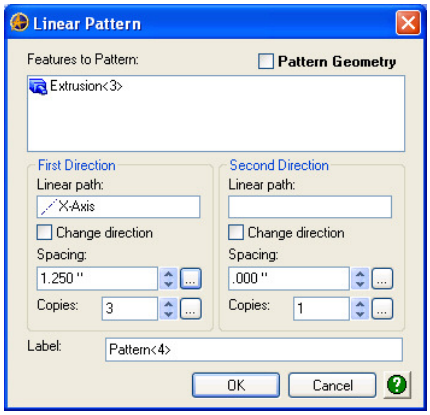
➢ ***To create a linear pattern:***

1. From the **Feature** menu, select **Pattern > Linear;** or, select the **Linear Feature Pattern** tool from the Part Modeling Toolbar. The **Linear Pattern** dialog appears.



2. Select the features to be patterned.

3. Check the **Pattern Geometry** box if you want to pattern the geometry exactly. You will normally be concerned with this option when you are patterning a feature that has been created using references to existing geometry such as “extrude to geometry." Below are two models. The first shows a pattern created with the Pattern Geometry box unchecked, and the second with the box checked. The cylinder was sketched on the ZX Plane, and extruded to the sloped face.



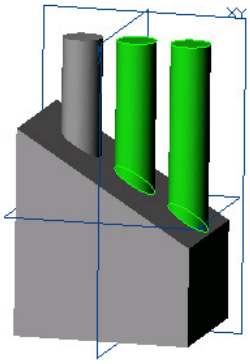
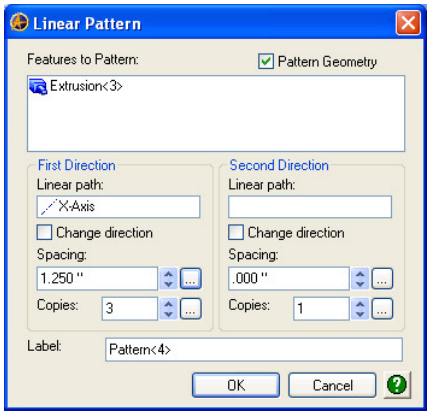


Figure 58: Example of Feature Patterned Without Pattern Geometry Option



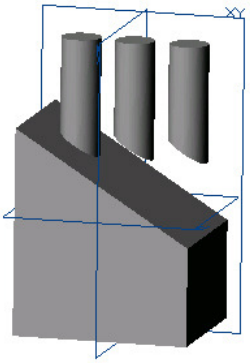


Figure 59: Example of Feature Patterned Using Pattern Geometry Option

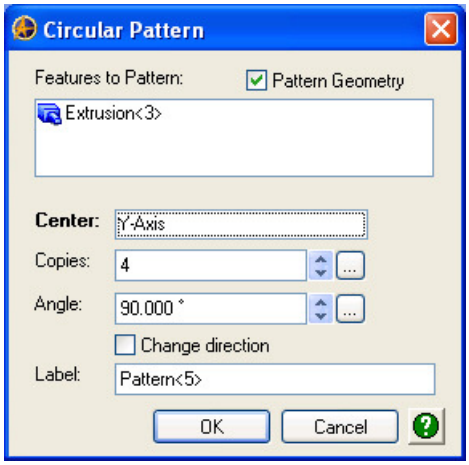
4. Select **Linear path** for the **First Direction**. An axis or linear edge can be used for the path.
5. Specify the number of **Copies** including the original feature.
6. Specify the **Spacing** between each copy.
7. Click the **Change direction** option to create the pattern in the opposite direction on the path.
8. If required, repeat steps **4-7** to create the pattern in a second direction as well.
9. Click **OK** to create the pattern.

Circular Patterns

You can use a circular pattern to repeat a feature in a radial direction around a centerline.

➢ ***To create a circular pattern:***

1. From the **Feature** menu, select **Pattern > Circular**; or, select the **Circular Feature Pattern** tool from the Feature Patterns fly-out on the Part Modeling toolbar. The **Circular Pattern** dialog appears.



2. Select the features to be patterned.

3. Check the **Pattern Geometry** box if you want to pattern the geometry exactly. See the previous section on Linear Patterns for more information on Pattern Geometry.

4. Select the **Circular path center**. An axis or linear edge can be used as the center.

5. Specify the number of **Copies** including the original feature.

6. Specify the **Angle** in degrees to control the spacing between the copied features.

7. If necessary, select **Change direction** to create the pattern in the opposite radial direction.

8. Click **OK** to create the pattern.

7.15.3 Topology Patterns

Topology patterns are similar to Feature Patterns, except that you do not have to have a feature to use them. Topology Patterns can be used to make copies of existing geometry, such as faces and edges. They are particularly useful when you are working with an imported model that does not have a feature history.

Note: You can pattern entire features using the Topology Pattern functionality, but it is better design practice to use Feature Patterns for that purpose. Also, it is important to remember that the entire created pattern must be on the same face as the original geometry, or an invalid body will result.

Topology Patterns vs. Feature Patterns

Feature Patterns and Topology Patterns will often provide similar results. However, the inputs for each type of pattern are different, as are the outcomes they can provide. Feature Patterns are used to pattern complete features, which means there must be a feature history in the Design Explorer. Models that are imported into Alibre Design do not have a feature history. Topology Patterns can be used to pattern actual geometry and do not require a feature, so they can be used on imported parts more easily.

Most of the time, you will use Feature Patterns. If you find that you can not accomplish your design goals with a Feature Pattern, try using a Topology Pattern.

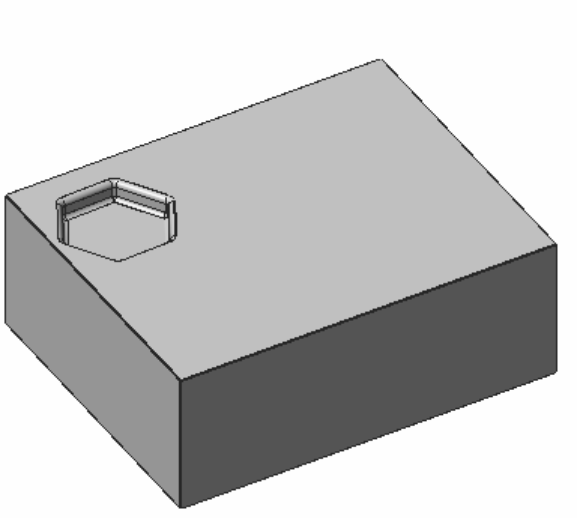
Pattern Inputs

- Feature Patterns require you to tell Alibre Design which features you want to pattern. You must pattern complete features.
- Topology Patterns require you to tell Alibre Design which faces you want to pattern. You can pattern any set of faces - either a set that would make a complete feature, or a set that represents only part of a feature.

Pattern Outcomes

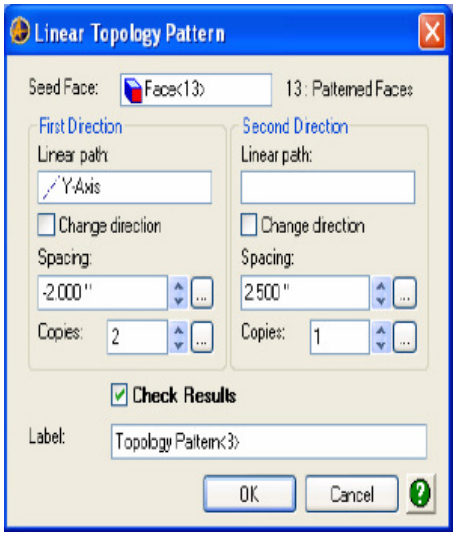
- Feature Patterns will result in a pattern of complete features.
- Topology Patterns will result in a pattern of sets of faces. These may represent a complete feature but may not. You should use a Feature Pattern if your design intent is to pattern the complete feature.

The following segments for creating Linear Topology Patterns and Circular Topology Patterns will demonstrate using this functionality to pattern a portion of existing geometry on this part that was imported into Alibre Design as a .sat file.



➢ ***To create a linear topology pattern:***

1. From the Feature menu, select Topology Pattern > Linear; or, select the Linear Topology Pattern tool from the Part Modeling Toolbar (available in the Feature Pattern fly-out). The Linear Topology Pattern dialog appears.



2. Select the **Seed Face**. Using the Seed Face, Alibre Design will infer a group of related faces (adjacent to each other) that will be included in the pattern. Selecting a different face as the Seed Face may or may not change the group of inferred related faces, depending on your model. The inferred faces will be highlighted in the work area, and the number of selected faces will be displayed in the dialog next to the Seed Face.

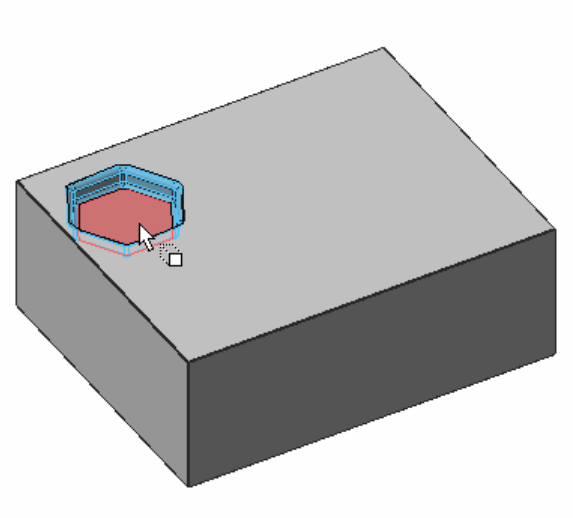


Figure 60: Model with Seed Face Selected

3. Complete all of the requirements for the First Direction. In **Linear Path**, select an edge or axis to determine the direction for the pattern creation. A preview will be displayed in the work area.

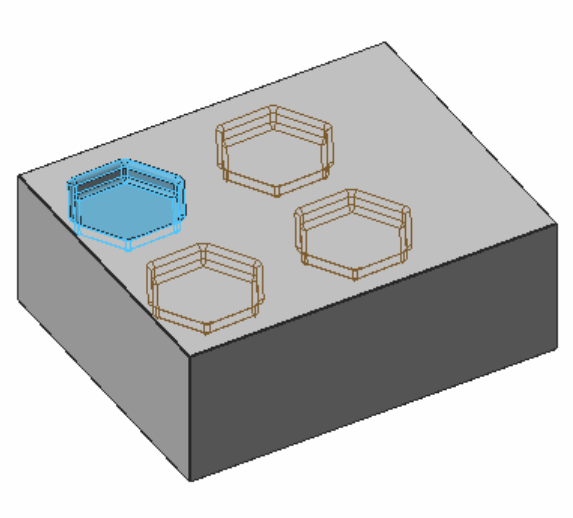


Figure 61: Linear Topology Pattern Preview

4. Check the **Change Direction** box to reverse the direction if necessary.

5. Enter the desired **Spacing** between each copy.

The resulting pattern cannot overlap itself. Setting the spacing too low can make the bodies overlap, resulting in an invalid body.

6. Enter the number of **Copies**, including the original.

7. If you would like to create the pattern in two directions, fill in the information for the **Second Direction** section as well.

8. Check the **Check Results** box to have Alibre Design detect whether or not the resulting bodies from the pattern are valid. It is good design practice to leave this box checked on. If this box is unchecked, you may inadvertently create a pattern that appears to be valid, but in reality is not.

   If you do not use the Check Results option, you should go to the Tools menu and select **Check Part** after creating the pattern to verify that your model is a valid body.

9. In **Label**, change the name of the pattern if desired.

10. Click **OK** to create the pattern.

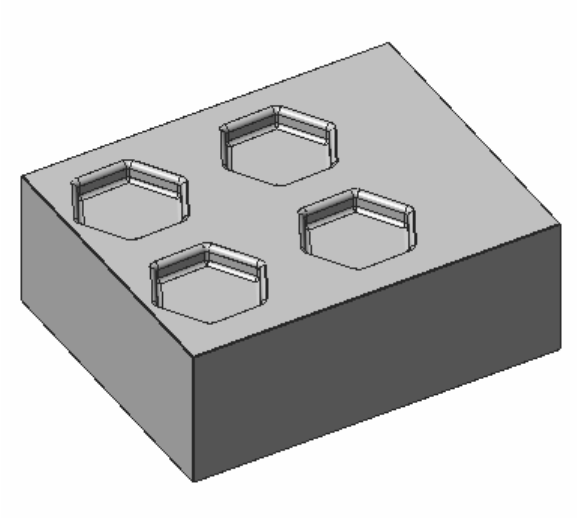
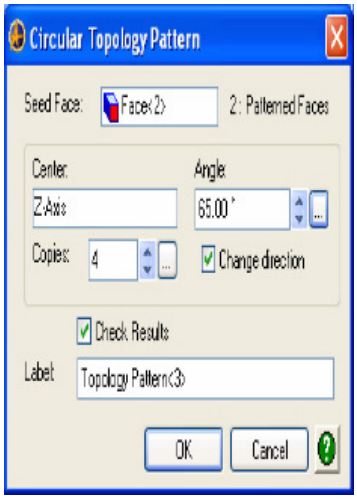


Figure 62: Linear Topology Pattern Completed

➢ ***To create a circular pattern:***

1. From the **Feature** menu, select **Topology Pattern > Circular**; or, select the **Circular Topology Pattern** tool from the Part Modeling Toolbar (available in the Feature Pattern fly-out). The **Circular Topology Pattern** dialog appears.



2. Select the **Seed Face**. Using the Seed Face, Alibre Design will infer a group of related faces (adjacent to each other) that will be included in the pattern. Selecting a different face as the Seed Face may or may not change the group of inferred related faces, depending on your model. The inferred faces will be highlighted in the work area, and the number of selected faces will be displayed in the dialog next to the Seed Face.

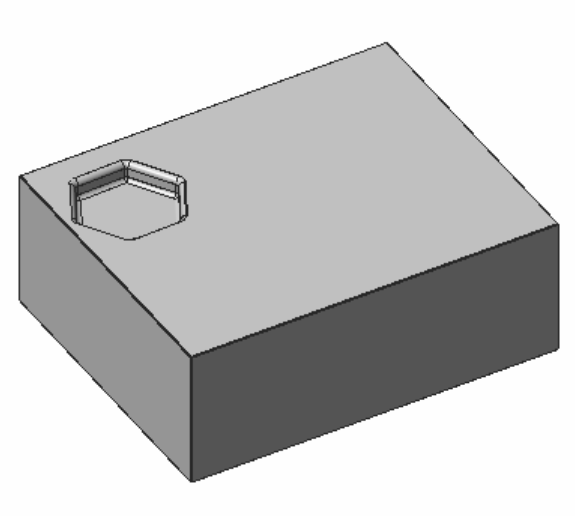


Figure 63: Circular Topology Pattern Model with Seed Face Selected

3. In **Center**, select an edge, axis, point, or cylindrical face for the axis the pattern will be created about. A preview will be displayed in the work area.

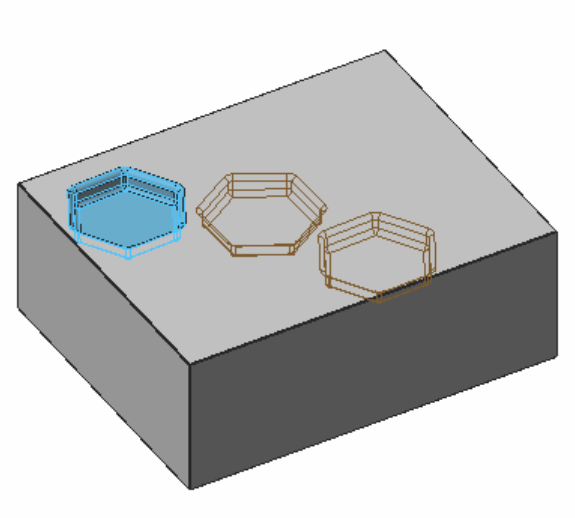


Figure 64: Circular Topology Pattern Preview

4. Enter the number of **Copies**, including the original.

5. Enter the desired **Angle** between each copy.

The resulting pattern cannot overlap itself. Setting the Angle too low can make the bodies overlap, resulting in an invalid body.

6. Check the **Change Direction** box to reverse the direction, if necessary.

7. Check the **Check Results** box to have Alibre Design detect whether or not the resulting bodies from the pattern are valid. It is good design practice to leave this box checked on. If this box is unchecked, you may inadvertently create a pattern that appears to be valid, but in reality is not.

If you do not use the Check Results option, you should go to the Tools menu and select Check Part after creating the pattern to verify that your model is a valid body.

8. In **Label**, change the name of the pattern, if desired.

9. Click **OK** to create the pattern.

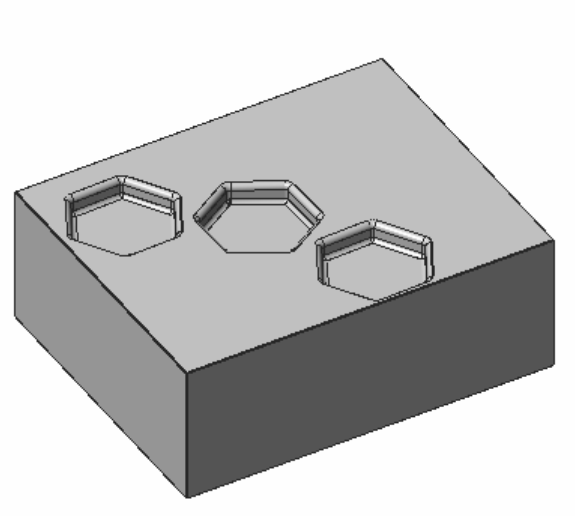


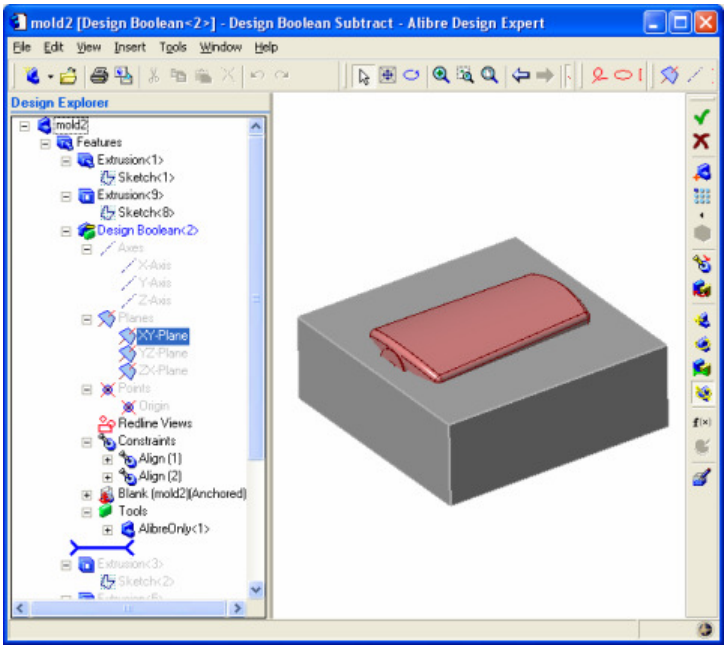
Figure 65: Circular Topology Pattern Completed

7.16 Design Boolean Features

You can use the Design Boolean Features to model parts for special applications such as packaging and mold design. This is accomplished by using a “tool” or “tools” (a collection of other parts or assemblies) to modify a “blank” (the current part). The creation of a Design Boolean is done in a special workspace called the Design Boolean Editor Environment. The Boolean feature is parametrically related to the tool part. Therefore, if the tool is modified and saved, the Boolean feature will update upon reopening. You can also modify the dedicated assembly that contains the blank and the tool.

7.16.1 Design Boolean Editor Environment

Upon selecting a Design Boolean Feature, the Blank part workspace transforms into the Design Boolean Editor Environment. This special workspace exists within a part workspace, but resembles an assembly workspace regarding toolbars and options. Inside the editor environment you will create a dedicated assembly that contains the blank part and the tool(s), constrained as required for modifying the design. The editor environment workspace is shown below.



There are two ways to exit the editor environment: you can commit or discard the Boolean feature. Once you choose one of those options, the workspace is transformed back into the blank part workspace. From the **Edit** menu, select Commit or Discard, or click the appropriate icon:

Commit

Discard

7.16.2 Creating Design Boolean Features

Although the Boolean Unite, Subtract, and Intersect features are different in end result, the steps used to create them are identical. The operations are the same as in set theory:

- Boolean Unite will add tool material to the blank part material.
- Boolean Subtract will remove tool material from the blank part material.
- Boolean Intersect will show the overlap of the tool material and the blank part material.

Before you begin:

You will need to have a part that you want to modify. This will be your “blank." In addition, you will need to have an existing part or assembly to use as a “tool." The tool must be saved to the file system.

➢ ***To create a Boolean design feature:***

1. Open the blank part.

2. From the **Feature** menu, select **Feature > Boolean > Subtract (or Unite, or Intersect)**. The **Insert Part/Subassembly** dialog appears.

   OR

   Select the appropriate tool from the Design Boolean Feature fly-out on the Part Modeling toolbar.

3. Highlight the part or assembly you want to use for the tool; then click **OK**. The part workspace window transforms into the Design Boolean Editor window. Click in the Editor workspace to place the tool. Click **Finish** when you have the number of copies desired.

   NOTE: You can repeat steps 2 and 3 to add as many parts or assemblies as required.

4. Constrain the tool(s) to the blank using the assembly constraints available in the Boolean editor workspace.

5. Click the **Commit** icon, or from the **Edit** menu, select **Commit** to complete the Boolean feature. The window transforms back into the blank part workspace, and the Boolean feature will be listed as a feature in the design explorer.

   NOTE: If a part or assembly in the tool is hidden in the dedicated assembly, it will not participate in the Design Boolean feature until it is unhidden.

7.16.3 Editing Design Boolean Features

The Boolean feature is parametrically related to the tool part or assembly. Therefore, if the tool is modified and saved, the Boolean feature will update upon reopening (if you make changes to the tool *inside the assembly context* (see "Editing a Part in an Assembly" on page 229), the changes will automatically update without saving and reopening). You can also modify the dedicated assembly that contains the blank and the tool by repositioning parts as well as adding or removing parts.

➢ ***To edit a Boolean design feature:***

1. In the Design Explorer, right-click the Design Boolean feature you want to modify.

2. Select **Edit**. The Design Boolean Editor Environment workspace appears.

3. Make the necessary modifications to the dedicated assembly (remembering that hidden parts and assemblies will not participate in the Design Boolean feature); then from the **File** menu, select **Commit** to apply the changes. The workspace will transform back into the part workspace.

7.17 Direct Editing

Direct Editing is the ability to manipulate a model without explicitly editing a feature or a sketch. Using the Direct Editing tools, you can move, remove, or resize existing geometry. Direct Editing tools are available in a part workspace, or when you are editing a part in the context of an assembly.

Direct Editing has several useful applications. You can use it to modify models that have been imported into Alibre Design, and therefore contain no design history (which means there are no features or sketches available to modify). You can use Direct Editing when you are designing a part in Alibre Design - there are instances when this would be easier than editing a feature or sketch explicitly. You can also use Direct Editing for conceptual design. You can move and resize parts of the model until it looks approximately like you want it, which gives you an idea of how the design will fit together. Then you can go back to the sketches and features and add precise dimensions to finalize the design.

Using the Direct Editing tools does not edit or delete any existing feature or sketch in the Design Explorer. It creates a new feature in the tree. Depending on which tool you use, the new feature may be an Offset Face, Move Face, or Remove Face feature.

7.17.1 Implications of Using Direct Editing

Direct Editing tools are not intended to replace feature creation tools, or the editing of features and sketches to achieve your desired results. They are intended to supplement the design process by enabling you to quickly modify the size of sketches and features on your model. You should use the traditional methods (such as editing a sketch or deleting a feature) for simple design changes.

Using Direct Editing can leave created dimensions as inaccurate representations of the new model. As an example, reference the images below. If you created an extrusion with a height of 2", you will have a model as shown in image A. You can perform a Push Pull Face operation to the top face (image B), and the resulting body will be as shown in image C. The new height of the extrusion is most likely what you are now interested in. The dimension of 2" is no longer an accurate representation of the height of the extrusion. The new total height is 2" plus the Push Pull Face dimension.

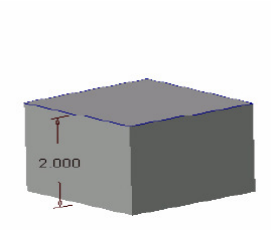


Figure 66: Original Extrusion

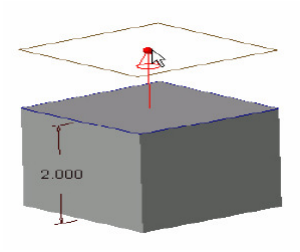


Figure 67: Performing Push Pull Face Operation

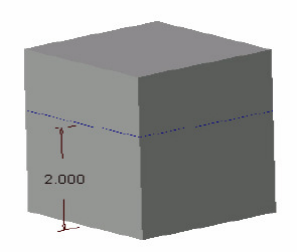


Figure 68: Final Model

Direct Editing and 2D Drawings

Because of this, 2D drawings will also be affected when you create drawings of models that have included Direct Edit functions. When you create a 2D drawing, in the Standard View Creation dialog, you have the option to include Design Dimensions. This option captures the dimensions you placed in the model and projects them into the drawing view. If you create a drawing of the block in the previous example, you will get a view similar to the one below. You can see the 2" dimension is located at the original height of the block. The 2" dimension is still the design dimension for that feature. In this case, if you want to represent the new height of the block, you would have to manually place a dimension - in this example it is the 3.185" dimension.

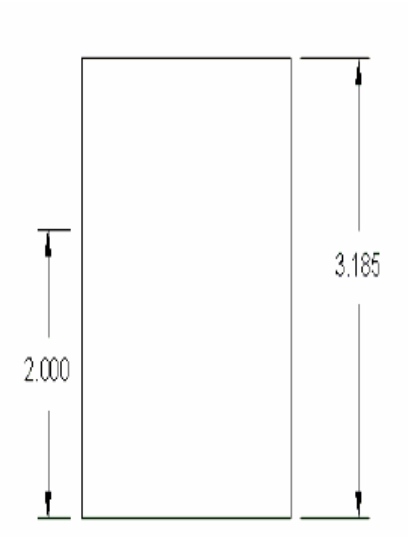


Figure 69: Drawing View of Block After Direct Editing

It is important to realize that you will need to manually add dimensions for most of the features that you modify using the Direct Editing tools. However, the Direct Editing tools can speed up the design process, particularly on imported models. The following sections describe each of the Direct Editing tools.

7.17.2 Push Pull Face Sketch

You can use the Push Pull Face/Sketch tool in two ways. First, you can use it to offset planar faces by a distance. Second, you can use it on a sketch that has not yet been used to create a feature, which will create an Extrude Boss feature (based on the selected sketch).

➢ ***To use the Push Pull Face tool:***

1. Select the **Push Pull Face/Sketch** tool from the Direct Editing fly-out on the Part Modeling toolbar; or, from the **Feature** menu, select **Direct Edit > Push Pull Face or Sketch**. The Push Pull Face overlay appears. You may need to maximize your workspace in order to view the entire overlay.

   The Push Pull Face overlay has two sections. If you have the "Show Full Overlay" option checked on, you will see both sections displayed. If you have this option checked off, you will see the top section. You can then display the bottom section by moving your mouse pointer over the overlay. To turn this option on and off, from the **Tools** menu, select **Options**. On the **Overlays** tab, check or uncheck **Show Full Overlay**. Or, you can click the **Pin** button to pin or unpin the bottom section.

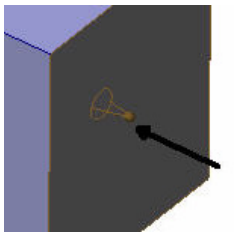
2. Click in the **Selection** field to activate it. Left click to select a planar face or an unconsumed sketch.

3. Check the **Snap Every** option if you would like the value of the Depth to snap at a defined increment.

4. Set the **Depth** to push or pull the face or sketch to. You can do this in two ways:

   a. Enter a value in the **Depth** field and select the **Refresh** tool (or press the Tab key) to see a preview.

   b. Left click on the "hot spot" shown on the face and drag it until you have it placed approximately where you want it. The Depth field will be populated with the new value.



5. Click **Reset** to erase the current item from the Selection field and set the Depth value back to 0.

6. Check the **Auto Apply** option on if you want Alibre Design to automatically apply the current option when you select a new face to perform an operation on. (If this option is unchecked, selecting a new face will begin a new operation without applying the current one.)

   The Auto Apply function only applies if you select a subsequent face or sketch to perform an operation on. If you use the tool on only one face or sketch, and then click Exit, the operation will not automatically be applied. So you can use Auto Apply when you are making multiple changes to multiple faces.

7. Click **Apply** to accept the new position. The model is updated, and a new feature is listed in the Design Explorer. The Push Pull tool remains active.

8. Left click another face or sketch to perform another Push Pull Face operation, or click **Exit** to leave the Push Pull mode.

Common Errors When Using Push Pull Face

1. Selecting a face that has fillets on it will cause the operation to fail. To fix this error, delete the fillet, then apply the Push Pull Face operation, and then place the fillet. In some cases you can suppress the fillet, apply the Push Pull operation, and then unsuppress it rather than deleting it.

2. Selecting a chamfer as the face or selecting any face that has other faces meeting it at angles other than 90 degrees can cause unexpected results. Depending on the results you want, it may be more beneficial to create an Extrude Boss feature instead of using Push Pull Face.

7.17.3 Push Pull Pocket or Boss

You can use the Push Pull Pocket or Boss tool to move a Pocket or a Boss by a distance in a linear direction. This tool allows you to do this without editing a feature or a sketch. It is useful for manipulating files that have been imported into Alibre Design and do not have features or sketches available to edit.

A Pocket is an area of the model where material has been removed from the solid body. This could be a Cut feature, but it does not have to be. The following example contains one large pocket, with 2 smaller pockets within it, for a total of 3 pockets.

A Boss is a solid portion of the model - it can be a feature or a part of a feature. It could be one of the Boss features such as Extrude Boss or Sweep Boss, but it does not have to be. In the following example, there are 4 separate Bosses. You can choose to move any combination of them with a Push Pull function.

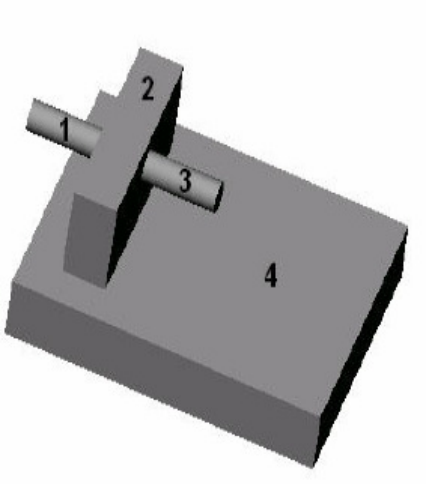


Figure 70: Model showing example of 4 separate bosses

➢ ***To use the Push Pull Pocket or Boss tool:***

1. Select the **Push Pull Pocket or Boss** tool from the Direct Editing fly-out on the Part Modeling toolbar; or, from the **Feature** menu, select **Direct Edit > Push Pull Pocket or Boss**. The Push Pull Pocket or Boss overlay appears. You may need to maximize your workspace in order to view the entire overlay.

   The Push Pull Pocket or Boss overlay has two sections. If you have the "Show Full Overlay" optn checked on, you will see both sections displayed. If you have this option checked off, you will see only the top section. You can then display the bottom section by moving your mouse pointer over the overlay. To turn this option on and off, from the Tools menu, select Options. On the Overlays tab, check or uncheck Show Full Overlay. Or, you can click the Pin button to pin or unpin the bottom section.

2. Check the **Infer Pockets** option on if you want Alibre Design to infer additional faces to include based on the selection you make. Uncheck it if you want to select all of the faces manually.

3. Click in the **Selection** field to activate it. Left click to select a face or combination of faces. You can select as many as necessary, but you must select a complete pocket or boss. For example, in the previous image, boss 3 has two faces that must be selected, while boss 2 has five faces that must be selected.

4. Click in the **Direction** field to activate it, then select a linear edge, axis, plane, or planar face.

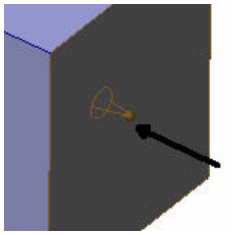
   Choosing an edge or axis allows you to move in one direction, while choosing a plane or planar face allows you to move in two directions.

5. Check the **Snap Every** option if you would like the value of the Depth to snap at a defined increment.

6. Set the **Depth** to move the pocket or boss. You can do this in two ways:

   a. Enter a value in the **Depth** field and select the Refresh tool (or press the Tab key) to see a preview.

   b. Left click on the "hot spot" shown on the face and drag it until you have it placed approximately where you want it. The Depth field will be populated with the new value.



7. Click **Reset** to erase the current item from the Selection field and set the Depth value back to 0.

8. Click **Apply** to accept the new position. The model is updated, and the Push Pull tool remains active.

The Auto Apply option only applies if you are using the Push Pull Face/Sketch tool.

9. Left click another face or sketch to perform another Push Pull Pocket or Boss operation, or click **Exit** to leave the Push Pull mode.

Common Errors When Using Push Pull Pocket or Boss

1. Selecting a face that has fillets on it can cause the operation to fail, if all of the affected faces are not selected. Even if you check the Infer Pockets option, some applicable faces may not be selected. To fix this error, visually inspect the area around the highlighted faces to determine if any faces have been excluded. If you find additional faces, select them.

2. If you move a pocket into free space (off of the solid body) it will disappear. This is because you are essentially moving a hole into empty space, where it can not exist.

7.17.4 Push Pull Radius

You can use the Push Pull Radius tool to offset cylindrical faces by a distance, thereby changing the radius. Using this tool creates an Offset Face feature in the Design Explorer. This tool is especially useful for manipulating imported files that do not have a Design Tree and consequently do not have features or sketches for you to edit.

➢ ***To use the Push Pull Radius tool:***

1. Select the **Push Pull Radius** tool from the Direct Editing fly-out on the Part Modeling toolbar; or, from the **Feature** menu, select **Direct Edit > Push Pull Radius**. The Push Pull Radius overlay appears. You may need to maximize your workspace in order to view the entire overlay.

The Push Pull Radius overlay has two sections. If you have the "Show Full Overlay" option checked on, you will see both sections displayed. If you have this option checked off, you will see the top section. You can then display the bottom section by moving your mouse pointer over the overlay. To turn this option on and off, from the Tools menu, select Options. On the **Overlays** tab, check or uncheck Show Full Overlay. Or, you can click the Pin button to pin or unpin the bottom section.

2. Click in the **Selection** field to activate it. Left click to select one or more cylindrical faces.

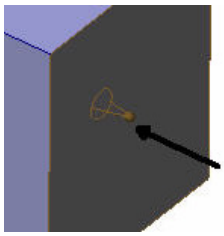
3. Check the **Snap Every** option if you would like the value of the Radius Offset to snap at a defined increment.

4. Choose the type of operation you want to apply by selecting Radius Offset or Final Radius from the drop down list. **Radius Offset** applies the designated offset value to each cylindrical face you selected. **Final Radius** is only available if you choose only one face, or if all of the faces you choose have an equal radius to start with. This option will change the value of the radius (or radii) to the designated value.

5. Set the new radius for the cylindrical face(s). You can do this in two ways:

   a. Enter a value in the **Final Radius To:** or **Offset Radius To:** field and select the **Refresh** tool (or press the Tab key) to see a preview.

   b. Left click on the "hot spot" shown on the face and drag it until you have it placed where you want it. The **To:** field will be populated with the new value.



6. Click **Reset** to erase the current item from the Selection field and set the **Radius To:** value back to 0.

7. Click **Apply** to accept the new position. The model is updated, and the Push Pull tool remains active.

The Auto Apply option only applies if you are using the Push Pull Face/Sketch tool.

8. Left click another face to perform another Push Pull Radius operation, or click **Exit** to leave the Push Pull mode.

Common Errors When Using Push Pull Radius

1. Some file types import cylindrical items as 2 separate faces, rather than one face. IGES is a common format that this occurs with. In these cases, you will notice that when you choose the cylindrical face, only half of it will highlight. If you click Apply, only half of the cylinder will resize. To correct this, you simply need to select the other half of the cylinder as well.

2. Using this tool on items that are not cylinders or cylindrical holes can cause unexpected results. Fillets are the most common example of this. You can use this tool on a fillet, but you may not get the result you were expecting. Below are examples of results you will see.

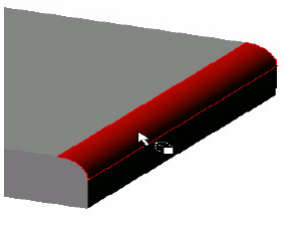


Figure 71: Model showing fillet selected for Push Pull Radius

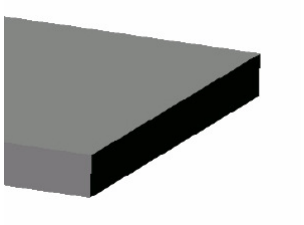


Figure 72: Model showing result of increasing radius on a fillet

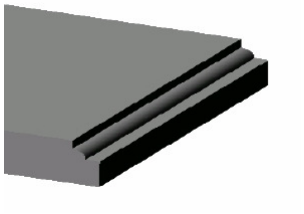


Figure 73: Model showing result of decreasing radius of fillet

7.17.5 Remove Faces

You can use the Remove Faces tool to remove design items from a model such as features, fillets, and portions of features. This operation does not delete any previously created feature or sketch - it creates a new Remove Face feature in the Design Explorer. This tool is especially useful for manipulating imported files that do not have a Design Tree and consequently do not have features or sketches for you to edit.

➢ ***To use the Remove Faces tool:***

1. Select the **Remove Faces** tool from the Direct Editing fly-out on the Part Modeling toolbar; or, from the **Feature** menu, select **Direct Edit > Remove Faces**. The Remove Faces overlay appears. You may need to maximize your workspace in order to view the entire overlay.

2. Check the **Use inferencing for:** option if you want Alibre Design to infer additional faces based on the selections you make. Then select either:

   a. **Pocket or Boss** - used for most selections except fillets; attempts to identify faces comprising a complete pocket or boss.

   b. **Fillet Chain** - attempts to identify a chain of fillets that are tangent to each other and have the same radius.

3. Select faces in the model. You can select as many faces as necessary. You can change the Inferencing options at any time (or uncheck it to turn it off) if necessary.

4. Click **Reset** to erase the current item from the Selection field.

5. Click **Apply** to accept the selection. The model is updated, and the Remove Faces tool remains active.

6. Left click another face to perform another Remove Faces operation, or click **Exit** to leave the tool.

Common Errors When Using Remove Faces

The most common error that will occur with this tool is not selecting everything necessary for the operation to succeed. If you select only a portion of a boss or pocket, you may get the error: "REM_NO_SOLUTION: gap cannot be filled". This typically means that Alibre Design does not know how the model should go together when the selected items are removed. To correct this, you need to visually inspect the selected items and determine which faces have been excluded. For example, complex fillets can sometimes leave a small face unselected. If this error occurs, pay close attention to the filleted areas of the model.

7.17.6 Tips for Successful Direct Editing

The best way to be successful with the Direct Editing tools is to practice using them so you will learn to identify situations where they will be useful while minimizing potential issues. These tools can help you accomplish many design goals easily, but there are situations where you should avoid using them.

- If you can easily edit the model using traditional methods, you should use them (such as by editing sketches or features).

  Direct Editing is not meant to be a replacement for making easy modifications. There are some modifications that are difficult or time consuming to make that can be accomplished easily with Direct Editing tools. In general, if you can edit a base sketch or feature quickly to get your desired result, you should.

- If you can delete a feature, do not use the Remove Tool.

  Using the Remove tool can be a quick way to remove features or portions of features. It is not intended to replace deleting a feature from the Design Explorer when doing that would accomplish your goal. For example, if you create a fillet that you later do not want, delete the Fillet Feature. Do not use the Remove Tool to remove the fillet.

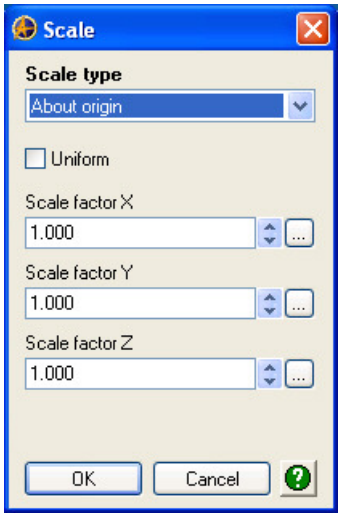
In general, these tips do not apply to the modification of imported geometry or conceptual design because the same potential issues will not typically arise.

7.18 Scaling Parts

You can increase or decrease the size of a part by using the Scale feature. Note that sheet metal parts cannot be scaled - only non-sheet metal solid parts. You can also use the Scale feature to mirror an entire part.

➢ ***To scale a part:***

1. From the **Feature** menu, select **Scale**. The Scale dialog appears.



2. In **Scale Type**, choose About Origin or About Centroid.

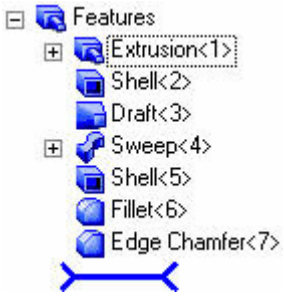
3. Check **Uniform** if you want to uniformly scale the part in all three directions. If you want a different factor for each direction, leave uniform unchecked and type in the desired values in Scale Factor X, Y, and Z boxes. A scale factor of 1.0 leaves the model unchanged.

   A feature is created in the design explorer when the scale operation is performed.

When modifying features before the scale operation, the model will roll back to the original dimensions.

7.19 Managing Features in the Design Explorer

After creating a feature, it will be listed in the Design Explorer under the **Features** node. Additional features will be listed in the order in which they were created. This feature order defines the part's construction history.



By right-clicking a feature in the Design Explorer you can:

- **Edit** the features properties
- **Suppress** the feature
- **Rename** the feature
- **Delete** the feature
- Check the **Status** of the feature

CHAPTER 8

Sheet Metal Feature Creation

Sheet Metal Part creation is not available in all versions of Alibre Design. Users of versions other than Alibre Design Professional and Expert can contact Alibre sales for information on adding Sheet Metal to your package. Like other solid parts, sheet metal parts are modeled by creating features. Features are individual 3D shapes representing common mechanical design elements, like tabs and cuts, which either create material or remove material in a part. Some features, such as tabs, require an associated sketch to define the 2D profile of the 3D shape. Other features, such as corner round and corner chamfer, can be created without a sketch and are applied to existing edges and faces.

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8.1 The Sheet Metal Part Modeling Interface

The Sheet Metal Part Modeling toolbar is shown by default on the right side of the workspace. Commonly used modeling tools are accessible on the Part Modeling toolbar.

Flat Tab ... create a flat tab feature

Flange ... create a flange feature

Closed Corner ... close a corner between two flanges

Formed Dimple ... create a dimple or drawn cutout

Cut ... create cut feature

Round Corner ... round a corner

Chamfer Corner ... chamfer a corner

Hole ... create a hole feature

Insert Catalog Feature ... insert a saved feature or sketch

Unbend ... unbend one or more flanges

Rebend ... rebend one or more unbent flanges

Flat Pattern ... view the model in a flattened state

Equation Editor ... open the Equation Editor

Generate to Last ... regenerate the part to the last feature to update changes

The tools that are accessible on the Sheet Metal Part Modeling toolbar are accessible from the **Feature** menu as well. The Feature menu also contains tools that do not have a corresponding toolbar icon.

Note: These features are all solid modeling features - using them in a sheet metal part may create a model that is incapable of unfolding to a flat pattern. Use with caution. Refer to *Feature Creation* (on page 101) for information on creating each of these features.

Features > Boss . . . create extruded, revolved, sweep, or loft boss features

Features > Cut . . . create extruded, revolved, sweep, or loft cut features

Features > Thin Wall Boss . . . create a thin wall extruded, revolved, or sweep boss feature

Features > Thin Wall Cut . . . create a thin wall extruded, revolved, or sweep cut feature

Features > Fillet . . . create a fillet feature

Features > Draft . . . create a draft feature

Mirror . . . mirror a feature about an edge or axis

Pattern > Linear . . . create copies of a feature in a linear pattern

Pattern > Circular . . . create copies of a feature in a circular pattern

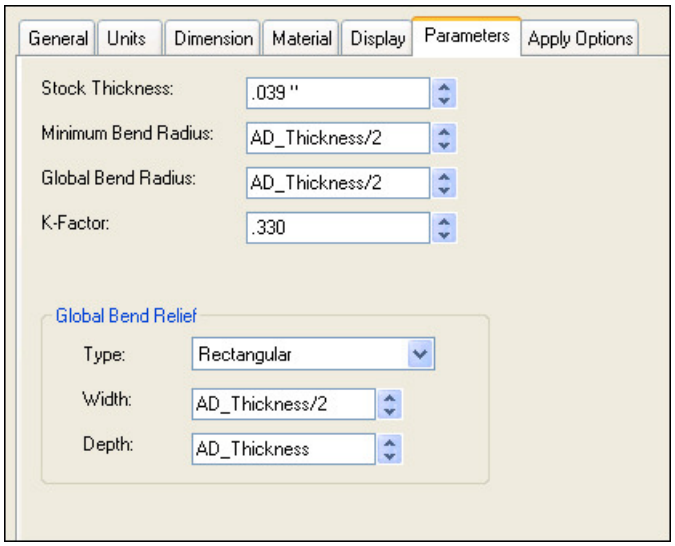
Save Catalog Feature ... save a feature for use in other models

8.2 Sheet Metal Part Parameters

Sheet metal parts have parameters that govern the way the part can be designed. The parameters are used as defaults in many sheet metal features. This allows a flat pattern to be resolved.

1. From the **File** Menu, select **Properties**. The Design Properties dialog appears.

2. Click the **Parameters** Tab.



3. Set your sheet metal material parameters of:

   - **Stock Thickness** - The thickness of the sheet of metal used to manufacture the part.
   - **Minimum Bend Radius** - The minimum Bend Radius to be allowed in the model. If you attempt to enter a radius smaller, it will automatically go to this minimum. Default Value = thickness / 2.
   - **Global Bend Radius** - The default bend radius used in all instances unless otherwise specified during feature creation. Default Value = thickness / 2.
   - **K-Factor** - During a forming process, "elongation" of the material occurs - a changing of shape due to the radius pushing material into another location. The "elongation" is called the K-Factor, or bend deduction. K-factors can be determined by using material charts.
   - **Global Bend Relief** - A small cut made in the material to prevent the bend radius from causing a distortion in the metal. Type can be **Rectangular** (produces a rectangle shape cut) or **Round** (produces a rectangular cut with a round on the short edge).

   Default values:

   - **Type** - rectangular
   - **Width** - thickness
   - **Depth** - thickness*2

4. Choose **Apply;** then **Close**.

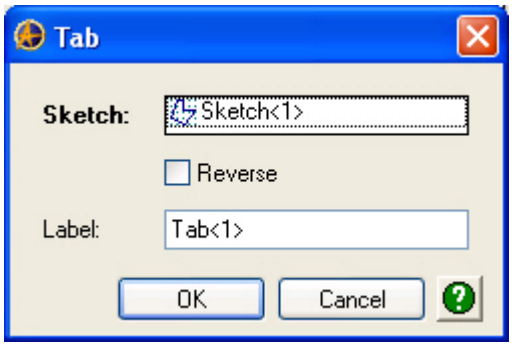
The parameters set here are accessible in the *Equation Editor* (see "Using Equations in Dimensions" on page 62).

8.3 Tab

A Tab is the first feature created in a sheet metal part, and inserts flat stock based upon a sketch. This is the only allowed initial feature. Tab thickness is obtained from the parameters set in the sheet metal part parameters. The name of the thickness parameter is "**AD_thickness**." The AD_thickness value corresponds to the stock thickness parameter specified in the Design Properties Parameters tab.

➢ ***To create a tab:***

1. Create a sketch and draw the profile you want to extrude.
2. From the **Feature** Menu, select **Tab.** Or, select the Tab tool from the Sheet Metal Modeling Toolbar. The Tab dialog appears.



3. Select the sketch you want to extrude.
4. Check **Reverse** if you want the Tab to project off the other side of the sketch plane from what the preview shows.
5. In **Label**, enter a unique name for this feature if desired.
6. Click **OK** to create the feature.

8.4 Flange

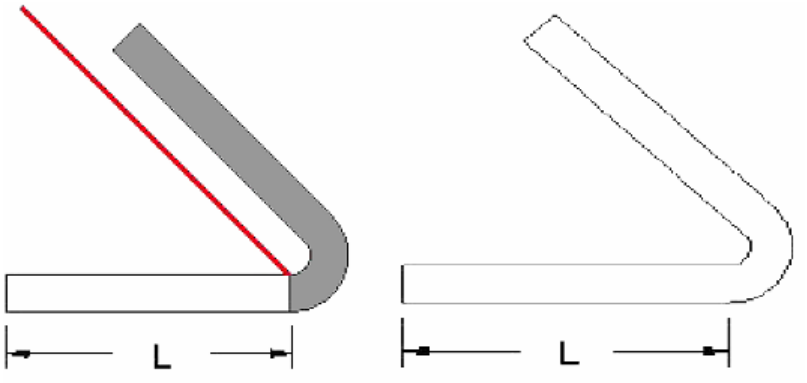
A flange is a section of the sheet metal that is bent from the original flat shape. You can add a flange to any straight layer edge of flat stock material. The flange places a bend and a rectangular (or trapezoidal for taper conditions) tab of flat stock material on the selected edge.

Please note that Flange features can not be mirrored.

➢ ***To create a flange:***

1. From the Feature menu, select Flange, or select the Flange tool from the Sheet Metal Modeling toolbar. The flange dialog appears.
2. On the Main Tab:
   a. In Edge, click the edge on the existing tab to which the flange will be attached.
   b. In **Alignment**, choose the **inside the tab wall**, **outside the tab wall**, or **adjacent to the tab wall** condition. Check the **Trim Side Bends** option, if needed.

      The following figures illustrate the difference in each of the alignment options. The solid line represents the sketch plane from which the flange is created, and the angle of the sketch plane (represented by the solid line here) is determined by the **Bend Angle** option described in step 7.



Note: Trim Side Bends can be checked only when the Bend Alignment Type is INSIDE or OUTSIDE. This option is applicable when the layer edge(s) adjacent to the selected edge have flanges. If this is true, then there will be setback part on the flat stock side, which would interfere with the bend on the adjacent edge. The distance by which the side bend is trimmed is equal to the setback of the current flange being made. If there are bends on both sides of the current flange being made, then both the bends are trimmed if this option is selected. Note that the side bend may be in the opposite direction also.

c. In Leg, type the length of your flange, and choose whether you want the length measured on the inside of the curve, the outside of the curve, or from the point where the bend ends. The length will be calculated as illustrated in the figures below.

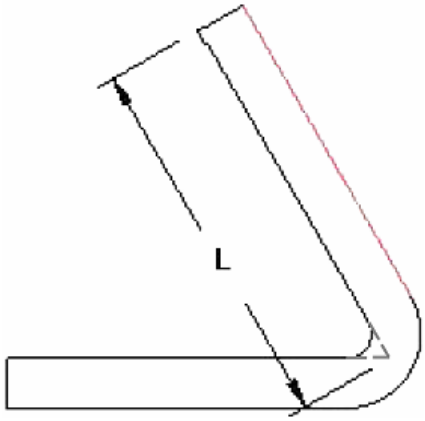


Figure 74: Flange Length Inside Option

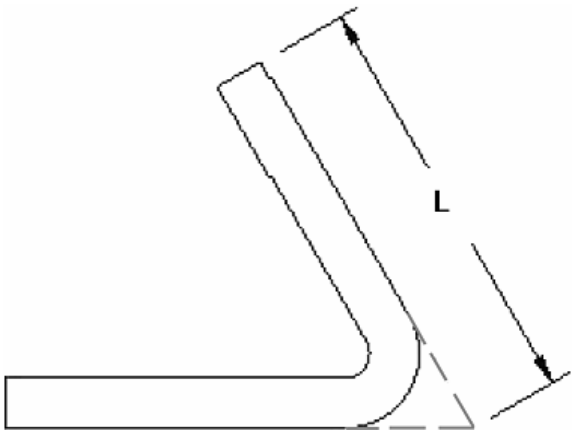


Figure 75: Flange Length Outside Option

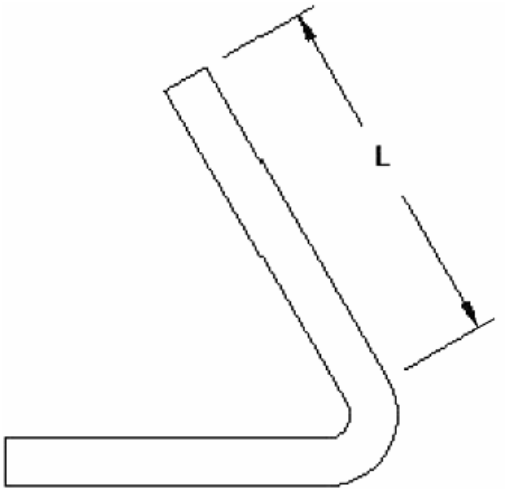


Figure 76: Flange Length Tab Option

d. Choose **Bend Only** if you do not want the flange wall, but just the bend that would be created for it. This option can be used for creating jogs.

e. Choose **Match Taper** if you want to specify that the taper of adjacent faces is continued through the flange.

f. In **Bend Angle**, type in the angle you want your flange with respect to the face you are adding it to. The bend angle is the "excluded" angle of the bend, measured as shown in the following:

Using a bend angle equal to or greater than 180° will disable inside/outside alignment options.

g. In **Bend Radius**, type the radius for your bend. This will be the inside radius of the bend. The default value here is the global bend radius specified in the parameters. Check **Reverse Bend** if you want the flange to go the opposite direction from what is showing in the preview.

3. On the Advanced Tab:

a. In **Bend Relief Type**, specify whether you want **No Bend Relief**, **Rectangular Relief**, or **Rounded Relief**.

b. If you choose to have a relief, enter the **Width** and **Depth**. (Default values are taken from the parameters set in the Properties dialog.)

c. **Bend Allowance – Choose Use K factor**: use the specified K-Factor to calculate allowance for the bend OR choose Use unfold length: set the unfold length used when unfolding this bend.

d. In **Label**, type in a unique name if desired.

4. Click OK to create the feature.

8.4.1 Sheet Metal Changes for Version 9.1 and Later

In version 9.1 of Alibre Design, modifications were made to the way flanges are created in sheet metal parts. These modifications ensure that the final dimensions of the sheet metal part are correct. They will affect the dimensions of the part and the dimensions of the flat pattern.

Your models will be affected in the following ways:

- Flanges that were created in a previous version of Alibre Design will continue to be generated using the old method, and the final dimensions of your model will remain the same.
- New flanges created in the model after opening the part in v9.1 or later will follow the updated process for creation. The final dimensions of your model will be correct, but may not behave as you expect, based on your past experience with the software.

It is possible to have flanges in your model that are calculated differently. This is because you can have flanges in your model that were created in a previous version, as well as new flanges created in version 9.1 or later.

How model dimensions are calculated

The tab length and flange length determine the overall dimensions of the final sheet metal part. The flange alignment type and length type that are selected during flange creation determines how the tab and flange length are measured in the part.

The new method of calculation will produce different dimension values than the old method when the following options are chosen:

- If the Bend Angle is anything other than 90 degrees, and the alignment type is Inside, the length of the tab feature will be different.
- If the Bend Angle is anything other than 90 degrees, and the length type is Inside or Outside, the length of the flange will be different.

Updating a model created in a previous version

You can update a part that was originally created in a previous version so that all flanges in the model are calculated using the updated method.

To do this:

1. From the **File** menu, select **Properties.**
2. Choose the **Parameters** tab. Check the box that says **Update all flanges in this part created in a version of Alibre Design prior to V9.1**.
3. Click **Apply**. You will be notified that the dimensions of the part will be permanently changed.

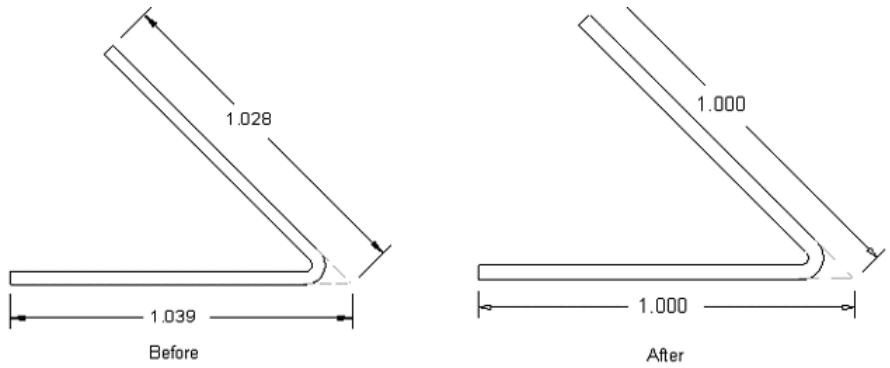
Once you have converted a model, it cannot be undone.

Here is an example of how the dimensions of the part will change:

Consider a 1”x 1” tab, on one edge of which we are adding a flange of length 1”. The options chosen are:

- Alignment type: Inside
- Length Type: Outside

The following figure shows the dimensions of the part before and after these changes have been implemented. The updated method shows the correct dimensions of 1.0" for both the tab length and the flange length.



8.5 Closed Corner

You can close corners where two bends meet in sheet metal bodies with appropriate extensions and square corner bend reliefs.

➢ ***To close a corner:***

1. From the **Feature** Menu, select **Closed Corner**. Or, select the Closed Corner tool from the Sheet Metal Modeling Toolbar. The Closed Corner dialog appears.
2. In **Edge**, click an edge that is part of the corner you want to close.
3. In **Join**, choose the method that you want to use to join the two flanges.

4. In **Label**, enter a unique name for this feature if desired.

5. Click **OK.**

8.6 Dimple

➢ ***To create a dimple:***

1. From the **Feature** menu, select **Dimple**. Or, select the Dimple tool from the Sheet Metal Modeling Toolbar. The dimple dialog appears.

2. In **Sketch**, select the sketch you would like to use to create the dimple.

Note: Sketches used to create dimples can contain only one closed figure.

3. In **Depth**, enter the depth of the dimple from the start face. The default is two times the thickness of the stock material.

4. In **Draft Angle**, enter the angle if needed. Only positive angle values are allowed. If a negative angle is entered, the **OK** button will not activate.

5. In **Sketch Alignment**, specify whether the tool profile sketch is to be applied to the outside of the dimple or the inside, which impacts the offset and draft angles used to create the feature.

6. Select **Cut Out Material** if you want to cut out the bottom of the dimple, resulting in a drawn cutout operation.

7. In **Include Rounding**, you can include automatic fillets for the dimple operation. **Die Radius** is the user specified radius for external fillets created by the dimple operation. **Punch Radius** is the user specified radius for internal fillets created by the dimple operation.

8. In **Round Profile Corners**, you can optionally round hard corners in the tool profile sketch.

9. In **Label**, enter a unique name for the dimple feature if desired.

10. Click **OK**.

8.7 Cut

Cutting removes material from a model based on the profile of a sketch. The cut will be punched through a single stock thickness of the sheet metal.

➢ ***To Create a Cut:***

1. From the **Feature** menu, select **Cut**. Or, select the Cut tool from the Sheet Metal Modeling Toolbar. The Cut dialog appears.

2. In **Sketch**, select the sketch you want to use for the cut.

Note: The sketch used in a cut must be placed on a sheet metal face.

3. In **Label**, enter a unique name for the cut feature, if desired.

4. Click **OK.**

8.8 Corner Rounds and Chamfers

You can create a round or chamfer on any corner of a sheet metal part.

8.8.1 Rounding a Corner

1. From the **Feature** Menu, select **Corner Round**. Or, select the Corner Round tool from the Sheet Metal Modeling Toolbar. The Corner Round dialog appears.
2. In **Items to Round**, select the edges you want to round. You may also select faces, which will round all corners adjacent to that face. Any combination of faces and edges may be selected.
3. In **Radius**, type the desired radius for the round. A preview will be displayed.
4. Enter the desired values in the distance fields. A preview will be shown of the chamfer.
5. Click **OK**.

8.8.2 Chamfering a corner

1. From the **Feature** Menu, select **Corner Chamfer**. Or, select the Chamfer Corner tool from the Sheet Metal Modeling Toolbar. The Corner Chamfer dialog appears.
2. In **Edges/Faces to Chamfer**, select the edges you want to chamfer. You may also select faces, which will chamfer all corners adjacent to that face. Any combination of faces and edges may be selected.
3. In **Chamfer Type**, select Distance-Distance, Angle-Distance or Equal-Distance.
4. Enter the desired values in the distance fields. A preview will be shown of the chamfer.
5. Click **OK**.

8.9 Holes

Alibre Design provides several standard holes. You can insert these standard holes on any planar face. The holes can be inserted to a specific depth ("blind"), through an entire model, or up to an intersection of a face. Holes are always inserted perpendicular to a face.

Refer to the Holes section in the solid Features Creation chapter for comprehensive information on *creating holes* (see "Holes" on page 125).

8.10 Unbend and Rebend

You can unbend and then rebend one or more flanges at a time. A feature is added to the feature tree in the Design Explorer, allowing you to insert other features, such as a cut, between an unbend and rebend pair.

- ***To unbend a flange:***

1. From the **Feature** menu, select **Unbend.** Or, select the Unbend tool from the Sheet Metal Modeling Toolbar.
2. In **Fixed Face/Edge**, choose a face or an edge to remain fixed throughout the operation.
3. In **Bends**, select the bends of the flanges to unbend, or choose **Select All Bends** to unbend all flanges.
4. Click **OK**.

- ***To rebend a flange:***

1. From the **Feature** menu, select **Rebend.** Or, select the Rebend tool from the Sheet Metal Modeling Toolbar.
2. In **Unbent Bends**, select the bends to rebend, or choose **Select All Unbent Bends** to rebend all the unbent flanges.

8.11 Catalog Feature

After features or sketches have been created, they can be *cataloged* (see "Saving Catalog Features" on page 127) and saved for use in other part models.

8.12 Copying Existing Features

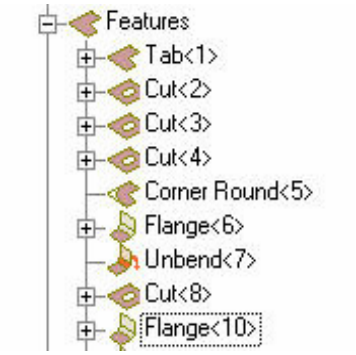
After sheet metal features have been created, many of them can be mirrored or patterned to easily create copies.

Please note that you cannot mirror Flange features. Flange features are dependent on the edge chosen during creation.

Refer to the Feature Creation chapter for comprehensive information on creating *patterns and mirrors* (see "Copying Existing Features" on page 128).

8.13 Managing Features in the Design Explorer

After creating a feature, it is listed in the Design Explorer under the **Features** node. Additional features are listed in the order in which they were created. This feature order defines the part's construction history.



By right-clicking a feature in the Design Explorer, you can:

- **Edit** the features properties
- **Suppress** the feature
- **Rename** the feature
- **Delete** the feature
- Check the **Status** of the feature

CHAPTER 9

Working with Parts

Parts are the fundamental component of a 3D design. Detail drawings are created from parts and assemblies are built by integrating multiple parts.

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Color Properties 165
Using the Project to Sketch Tool 165
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3D PDF Publishing 174

9.1 Using the Design Explorer

The Design Explorer on the left side of a workspace provides an outline of the part's design history and structure. The Design Explorer lists all features, sketches, faces, edges, vertices, planes, axes, points, section views, and redline views associated with a design.

The Design Explorer is displayed by default. You can hide the Design Explorer, although it is recommended that you keep it displayed when working. To hide the Design Explorer, from the **View** menu select **Design Explorer**. A check mark next to a menu item indicates the item is currently displayed.

Numerous tasks can be accomplished from the Design Explorer:

- Moving the cursor over an item or selecting an item in the Design Explorer will subsequently highlight or select the corresponding item in the work area.
- Right-click a feature to edit its properties, rename the feature, delete a feature, suppress a feature, and check the status of a feature.
- Drag and drop features in the Design Explorer to reorder them. Changing the feature generation order changes the construction of the part.
- Click the plus ⊞ or minus ⊟ signs next to an item to expand or contract an item or item group. For example, click the plus ⊞ sign next to a feature to see its associated sketch.

9.2 Modifying a Part

You can easily alter a part by editing sketches and features, suppressing features, reordering feature construction, and rolling back features.

9.2.1 Editing Sketches and Features

After the initial creation of a feature, you can always edit the original feature sketch or the defining properties of the feature.

➢ ***To edit a feature sketch:***

1. In the Design Explorer, click the plus ⊞ sign next to the feature to expand it and display its associated sketch.

2. Right-click the sketch and select **Edit** from the pop-up menu; or double-click the sketch.

Or

1. Select the feature in the work area.
2. Right-click and select **Edit Feature Sketch** from the pop-up menu.

   Or

   Double-click the feature in the work area to edit the sketch associated with that feature.

The feature sketch appears in sketch mode and you can edit the sketch.

➢ ***To edit a feature:***

In the Design Explorer, right-click feature and select **Edit** from the pop-up menu.

Or

Select the feature in the work area; then right-click and select **Edit Feature** from the pop-up menu.

The dialog associated with the feature type appears displaying the original feature properties. You can change the properties as required.

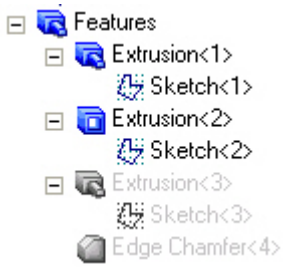
9.2.2 Suppressing Features in Parts

You can suppress features to temporarily remove them from a part. Suppressing features hides them in the display and prevents the feature from being used in any other modeling operations. Suppressed features are not calculated during regeneration.

➢ ***To suppress a feature:***

Right-click the feature in the Design Explorer and select **Suppress** from the pop-up menu.

The feature becomes hidden in the work area. The feature is listed in gray in the Design Explorer and the feature icon is shown in gray. Note that if you suppress a feature, features listed below it in the Design Explorer may fail, if they depended on the suppressed feature for successful regeneration.



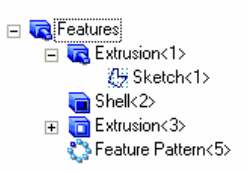
To remove the suppress state, right-click the suppressed feature and select **Suppress** from the pop-up menu.

9.2.3 Reordering Features

You can reorder features in the Design Explorer to alter the sequence in which features are created. Reordering features will change a part's construction.

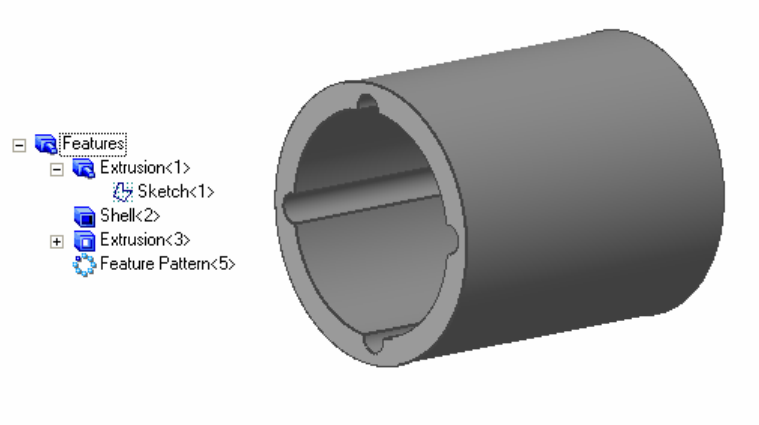
➢ ***To reorder features:***

1. In the Design Explorer, click and drag a feature up or down in the feature list. When you drag a feature in the Design Explorer the cursor will change.

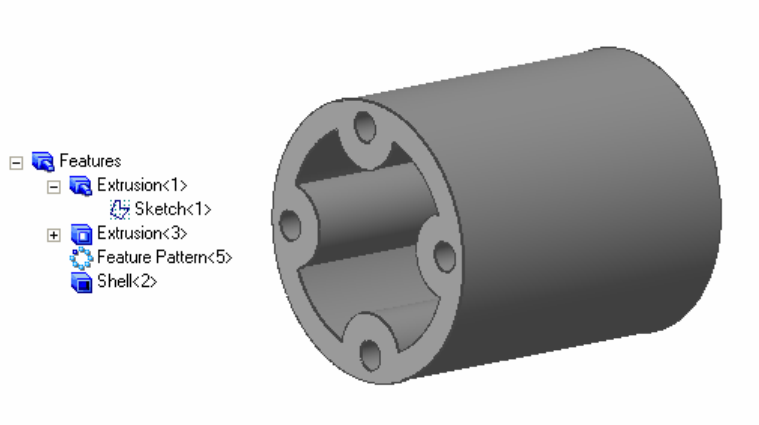


2. Release the mouse button when the feature has been dragged to the appropriate location in the feature list. The part will regenerate to reflect the new construction order.

Original feature list:



After reordering features:



9.2.4 Rolling Back Features

You can roll back a part to an earlier state in the design. When you roll back to an earlier state, features below the rollback point become inactive. When a part is rolled back, new features are inserted at the rollback point, not at the end of the feature list.

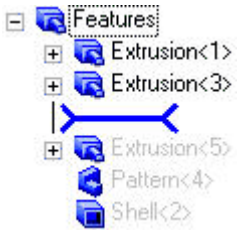
➢ ***To roll back features in the Design Explorer:***

1. In the Design Explorer, move the cursor over the blue **Feature History** line.
2. Click and drag the Feature History line above the features you temporarily want to disable.
3. Release the mouse button.

   Or

   Double-click the feature listed immediately above the features you want to disable.

The Feature History line moves to the rollback point, the features below the line become gray, and the part is rebuilt to only reflect the features above the Feature History line.



➢ ***To roll features forward:***

Drag the **Feature History** line below the feature or features you want to reactivate.

Or

Double-click the feature that you want to reactivate. All inactive features above the feature you double-click will also become active again.

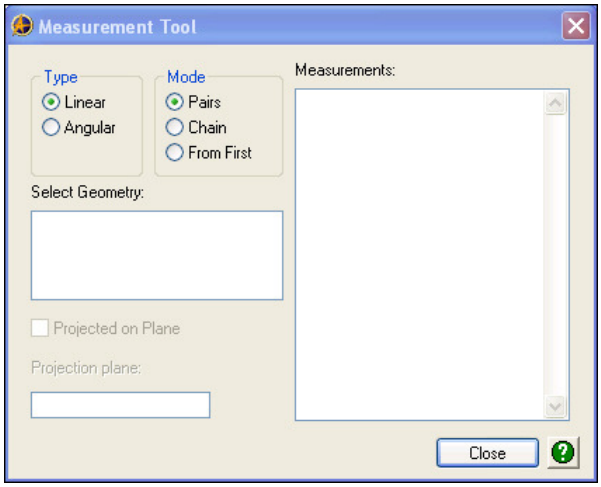
Note: To simultaneously reactivate all disabled features, select the **Regenerate** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Regenerate**; or press **F5** on the keyboard.

9.3 Using the Measurement Tool

You can use the measurement tool to measure items, as well as distances and angles between edges, faces, vertices, planes, axes, and points in a design.

➢ ***To take a measurement:***

1. Select the **Measurement** tool from the Inspection toolbar; or from the **Tools** menu, select **Measurement Tool**; or press **Ctrl + M** on the keyboard. The Measurement Tool dialog appears.



2. Select a measurement **Type**, either **Linear** or **Angular**.

3. Select a **Mode**:

   Pairs: measurement is taken between two entities, e.g. distance from edge to edge.

   Chain: cumulative measurement is taken between sequential selections, e.g. distance along multiple edges.

   From First: measurement is taken between two entities, with first selection always remaining constant.

4. Select an entity to measure or an entity to measure from.

Note: You can also pre-select entities and then open the Measurement Tool dialog. When pre-selecting, you must hold the Shift key down to select multiple items.

5. If applicable, select an entity to measure to.

Note: To measure to or from the center of cylindrical shapes, select the associated cylindrical face.

Need new graphic

6. Select the **Projected on Plane** option to project the measurement to another plane. Select the **Projection plane**.

   A measurement preview line is displayed in the work area and measurement information is displayed in the **Measurements** area in the dialog.

7. New measurements can be taken without closing the dialog. Adjust the **Type** and **Mode** as necessary and select the new entities.

8. Click **Close** when finished.

9.4 Part Display Options

You can select from four different display modes to control how parts are displayed.

The default display type is **Shaded**. Other display options include **Wireframe**, **Shaded & Visible Edges**, and **Shaded & All Edges**.

➢ ***To change the display:***

From the **View** menu, select **Display** and one of the four options**:**

- **Shaded:** displays parts in shaded mode, edges are not outlined.
- **Wireframe:** displays parts in wireframe mode, only edges are outlined and displayed.

Note: When viewing the display in wireframe, you can turn the silhouette edges of the model on or off. To do this, from the **View** menu, select **Display**. Check the **Silhouette Edges** option to turn them on. You can also use the tool to toggle them on and off.

- **Shaded & Visible Edges:** displays parts in shaded mode, only visible edges are outlined.
- **Shaded & All Edges:** displays parts in shaded mode, visible as well as hidden edges are outlined.

The display is updated.

Note: You can quickly change between shaded and wireframe display modes by selecting the **Shaded** and **Wireframe** tools from the Options drop down list on the Visibility toolbar.

9.5 3D Section Views

You can insert 3D section views into a part and subsequently take measurements on the sectioned face, or use the sectioned face in modeling operations.

➢ ***To insert a 3D section view:***

1. Select the **Insert 3D Section View** tool from the Inspection toolbar; or from the **Insert** menu select **3D Section View**; or right-click and select **Insert 3D Section View** from the pop-up menu. The **3D Section View** dialog appears.

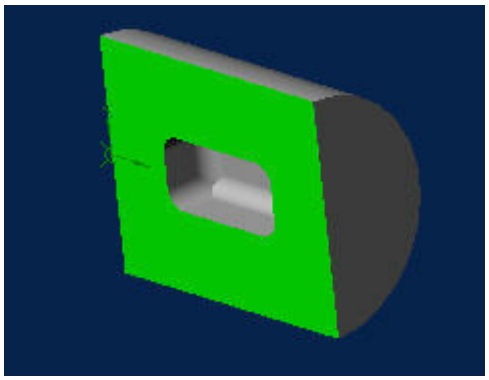
2. Click in the **Parts to Section** field. Select the part to be affected by the section.

3. Select a **Slicing Plane**. A reference plane or planar face can be used. A plane with arrows appears representing the cut.

4. If necessary, select **Reverse** to change the direction of the section view. The slicing plane points to the section side that will remain visible.

5. Enter a value in the **Offset** field to shift the cutting plane and create the section view the specified distance from the slicing plane.

6. Click **OK** to create the section view. The section view is also listed in the Design Explorer under the **Section Views** node.



➢ ***Managing 3D section views:***

You can turn section view visibility on and off as well as delete section views from the Design Explorer. To hide or display a section view, right-click the section view in the Design Explorer and select **View** from the pop-up menu. A check mark next to **View** indicates the view is currently displayed.

To delete a section view, right-click the section view in the Design Explorer and select **Delete** from the pop-up menu.

You can insert multiple section views into a part; however, you can only display one section view at a time.

9.6 Part Physical Properties

You can calculate and display the volume, mass, surface area, center of mass, inertial tensor, and principal axes of inertia of a part or assembly.

➢ ***To set a density value:***

1. From the **File** menu, select **Properties**. The **Design Properties** dialog appears.

2. Select the **Material** tab.

3. In the **Material** field, select the desired material from the drop down menu, or select **Custom** to enter your own density value. (The density is only valid for a part, not an assembly.)

4. Click **Apply** and then **Close**.

➢ ***To calculate physical properties:***

1. Select the **Physical Properties** tool from the Inspection toolbar; or from the **Tools** menu select **Physical Properties**. The **Physical Properties** dialog appears.

2. Select an **Accuracy** setting.

3. Click **Calculate**. The physical properties are displayed. If necessary, you can copy the physical properties information from the dialog and paste it into another document. The units displayed for these calculations can be set from the **File** menu by selecting **Properties**, and choosing the **Units** tab.

4. Click **Close** to close the dialog.

9.7 Color Properties

You can add color to a part as well as control a part's reflectivity and opacity.

➢ ***To apply color properties:***

1. Right-click in the work area and select **Color Properties** from the pop-up menu; or from the **Edit** menu select **Color Properties**. The **Color Properties** dialog appears.

2. Select the **Color** button to apply a color to the part, or the **Edge Color** button to apply a color to the part edges. The **Color** dialog appears.

3. Select a color from the **Basic colors** area or create a **Custom** color. Click **OK** and the preview will update to reflect your selection.

4. Click **OK** to close the Color dialog.

5. To add **Opacity** or **Reflectivity**, slide the controls appropriately. (Less opacity will make the part appear transparent, as glass or clear plastic. More reflectivity will make the part appear shiny, as metal or plastic.)

6. Click **OK** to apply the settings.

9.8 Using the Project to Sketch Tool

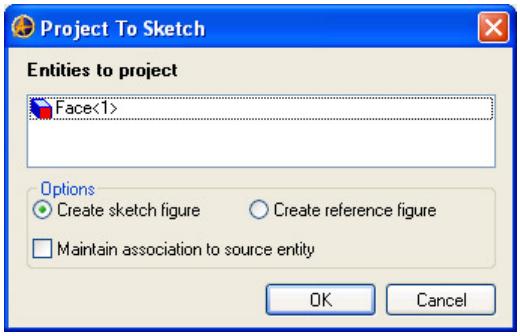
You can create new sketch or reference figures automatically by projecting existing edges onto a sketch plane. This function is useful when you are creating a new part (or modifying an existing part) in an assembly workspace. You can project edges from other parts in the assembly to the sketch in order to reference them in the new part.

➢ ***To use Project to Sketch:***

1. Select a sketch plane.

2. Enter sketch mode.

3. Select the **Project to Sketch** tool from the Sketching toolbar; or from the **Sketch** menu select **Project to Sketch**. The **Project to Sketch** dialog appears.



4. Select the existing feature edges that you want to project to the sketch plane.

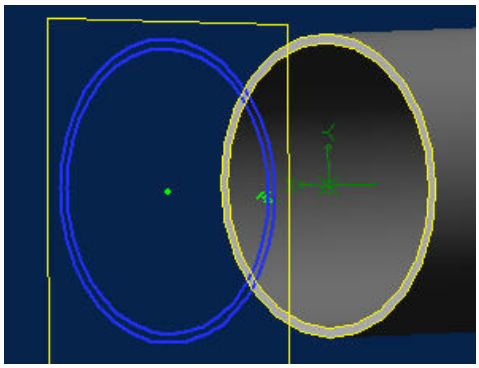
Note: You can select all the edges on a face at once by selecting a face.

5. To create a new sketch figure, select the **Create sketch figure** option.

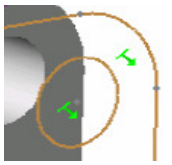
6. To create a new reference figure, select the **Create reference figure** option.

7. Select the **Maintain association to source entity** option if you want the new sketch or reference figure to reflect any changes made to the originating profile.

8. Click **OK** to create the new sketch or reference figure.

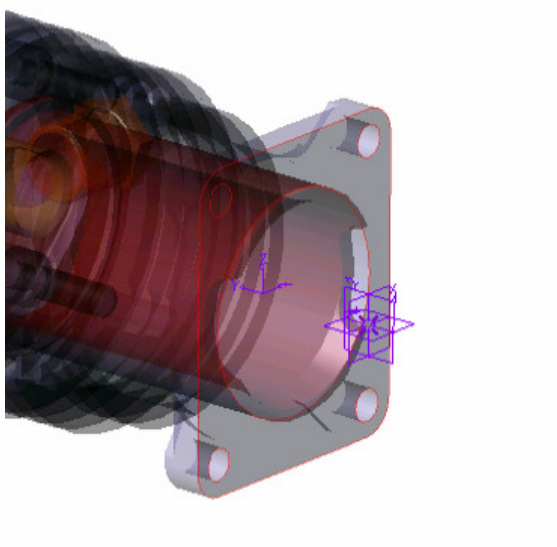


If you selected the **Maintain association to source entity** option, project-to-sketch constraint symbols are displayed on the new figures to indicate that they are constrained to the originating profile.

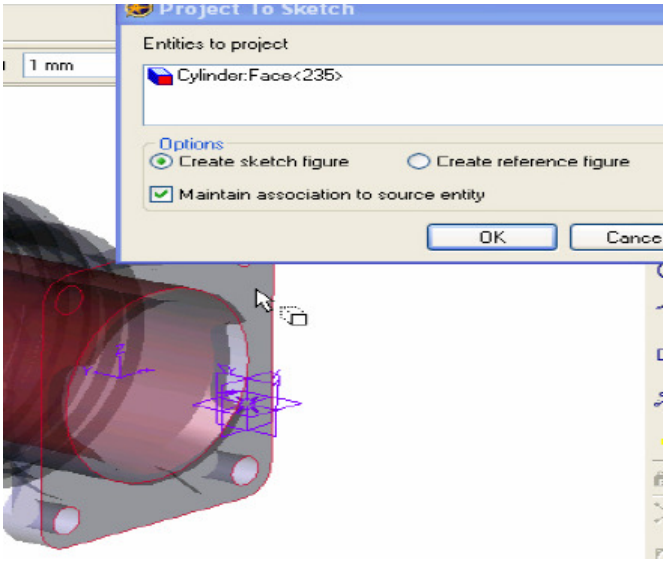


You will not be able to place dimensions on figures that were created with the Project to Sketch **Maintain association to source entity** option. You can, however, delete the constraints and subsequently add driving dimensions to the figure.

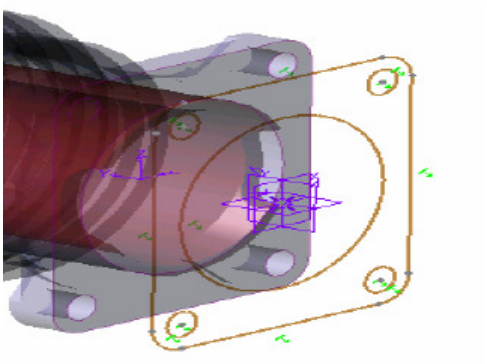
The following images demonstrate using Project to Sketch to create a new part in an assembly workspace. The assembly is shown slightly transparent in the background, and the reference planes for the new part are visible. The assembly is currently in Edit Part mode on the new part.



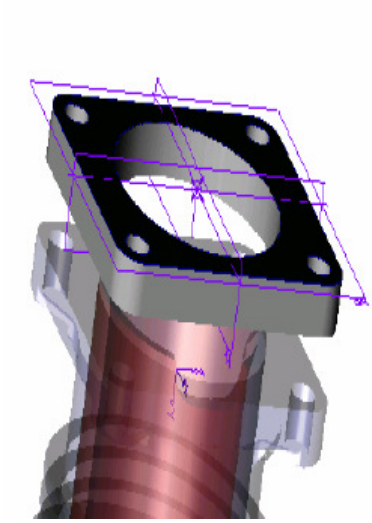
The Project to Sketch dialog is open, and the face of an existing part is selected. This will allow creation of a new part that will fit directly onto the existing face, with mounting holes in the correct location. Maintain association to source entity is selected, so if the existing face is modified, the new part will update as well.



Upon clicking the OK button, all of the edges of the selected face are projected onto the new sketch plane.



An Extrude Boss feature is created from the sketch.



The assembly is put back in Edit Assembly mode, and the new part can be constrained as necessary.

9.9 Spreadsheet Driven Designs

You can create variations of a design by using a spreadsheet of parameters. Additionally, a single spreadsheet of parameters can be shared by multiple designs. The spreadsheet must be created in Microsoft Excel; Excel 2000, Excel 2007 and Excel XP are supported.

Before you can drive a design with a spreadsheet of parameters, you must first set up Excel by installing the Alibre Design Add-In in Microsoft Excel. Next, you may either create a design with a desired set of dimensions, or create a spreadsheet first with a set of varying parameters for those dimensions. Then you are ready to drive the design with the spreadsheet.

Note: When using Excel 2007, you must be running Excel as Administrator. If you are not, you will receive an error message.

9.9.1 Setting Up Excel to Drive Designs

Before you can drive a design with a spreadsheet of parameters, you must first install the Alibre Design Add-In to Microsoft Excel.

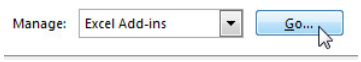
➢ ***To set up Excel 2000 or XP:***

1. Launch Alibre Design.
2. Launch Microsoft Excel.
3. From the **Tools** menu in Excel, select **Add-Ins**. The Add-Ins dialog appears.
4. Click **Browse**. The Browse dialog appears.
5. Browse to **Program Files > Alibre Design** on your PC's local C:/ drive and locate the file named “Alibre Design Add-In.xla.” (This is the default location for Alibre Design files at installation.)
6. Click the file name.
7. Click **OK**. The Alibre Design Add-In appears in the Add-Ins dialog with a checkmark next to it.
8. Click **OK**. In Excel, a new item appears in the Tools menu: **Alibre Design Add-In > Control Parameters.**

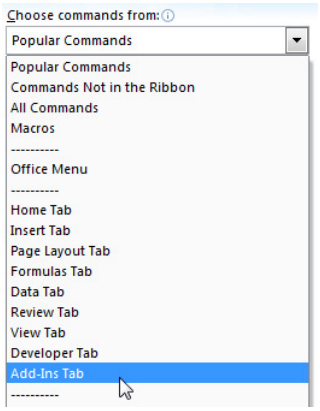
➢ ***To set up Excel 2007:***

1. Launch Microsoft Excel.
2. Click the Microsoft Office logo in the top left corner of the spreadsheet.

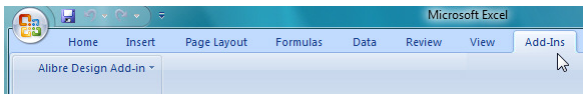
3. In the dialog, click the **Excel Options** button.
4. In the Excel Options dialog, choose **Add-ins**.
5. In the Add-ins section, in the Manage field, select **Excel Add-ins** from the drop down list, then click the **Go** button.



6. Click the **Browse** button and browse to **Program Files > Alibre Design** on your PC's local C:/ drive. Open the file named “Alibre Design Add-In.xla.” (This is the default location for Alibre Design files at installation.)
7. Make sure **Alibre Design Add-In** is checked, then click **OK**.
8. Click the Microsoft Office logo again.
9. In the dialog, click the **Excel Options** button.
10. Click **Customize** and expand the **Popular Command** drop down list.



11. Select **Add-ins Tab**.
12. Click **OK**.
13. The Add-ins tab is added.



9.9.2 Driving Designs by Spreadsheet

You can use Microsoft Excel spreadsheets to store and manage design information for use in driving one or more designs in Alibre Design. You may either set up the spreadsheet first, or design the part first.

TIP: You must go through the steps to *set up Excel* (see "Setting Up Excel to Drive Designs" on page 168) before you can drive a design.

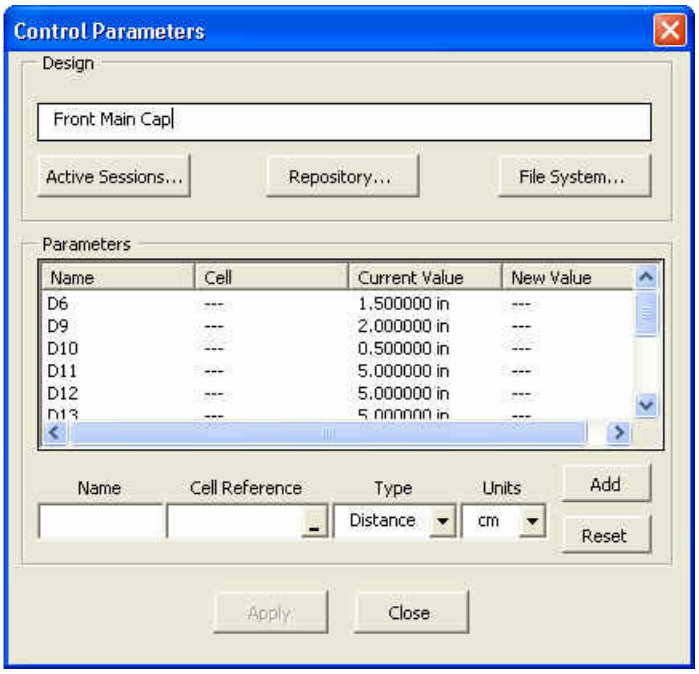
➢ ***To drive a design by spreadsheet:***

1. Launch Alibre Design.
2. Open the desired part.
3. Launch Microsoft Excel.
4. Save the workbook (spreadsheet).
5. Enter the desired settings for the parameters. For each parameter, enter a value, type, and units. You may also enter a name and comments.
6. From the **Tools** menu in Excel, select **Alibre Design Add-In > Control Parameters**. The Control Parameters dialog appears. The part name is displayed in the Design field and its dimensions appear in the parameters table.

   Or

   For **Excel 2007**, select the **Add-Ins** tab. Choose **Alibre Design Add-In > Control Parameters**.

Note: When using Excel 2007, you must be running Excel as Administrator. If you are not, you will receive an error message.



7. In **Parameters**, click one of the parameters. Its name appears in the Name field.

8. For **Cell Reference**, click the field, then click the cell on the Excel spreadsheet that the parameter should reference.

   Or

   Click the button. The cell reference box appears. Click the desired spreadsheet cell; then, click the button to return to the Control Parameters dialog.

9. In **Type**, select the parameter type: distance, angle or count.

10. In **Units**, set the desired units of measurement (for distance and angle parameters).

11. Click **Modify**.

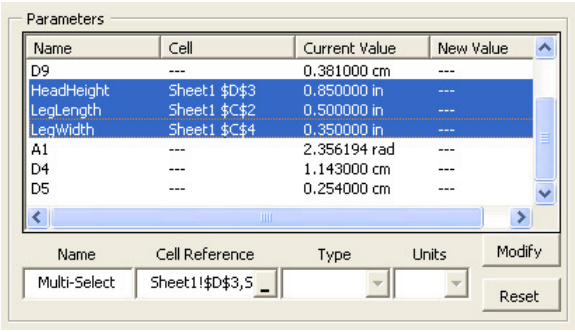
12. To add a new parameter, click **Reset**. The entry fields are cleared. In **Name**, type a name for the new parameter then follow steps 7-11 above.

13. Click **Close**. The open part in Alibre Design is modified to the new parameters.

➢ ***To set more than one parameter at a time***

You can select more than one parameter at a time in the Control Parameters dialog.

1. Select the first parameter you want to set, then hold the Ctrl key down and select any other parameters to modify. Each of the parameters will highlight.



   In the Name field, it will say Multi-Select and in the Cell Reference field, each of the cells will be listed in the order they appear in the Parameters section. (It will say NULL in the place of a cell name if no cell is currently linked.)

2. Click the button to move to the linked Excel spreadsheet, and select, in the correct order, the new fields that you want to link. You must click the same number of cells as parameters you highlighted.

3. Return to the Control Parameters dialog by clicking the button.

4. Click the Modify button to update the parameter values.

9.9.3 Modifying Spreadsheet Driven Parameters

Modify the parameters for spreadsheet driven designs directly in Microsoft Excel.

➢ ***To modify spreadsheet driven parameters:***

1. Launch Alibre Design.

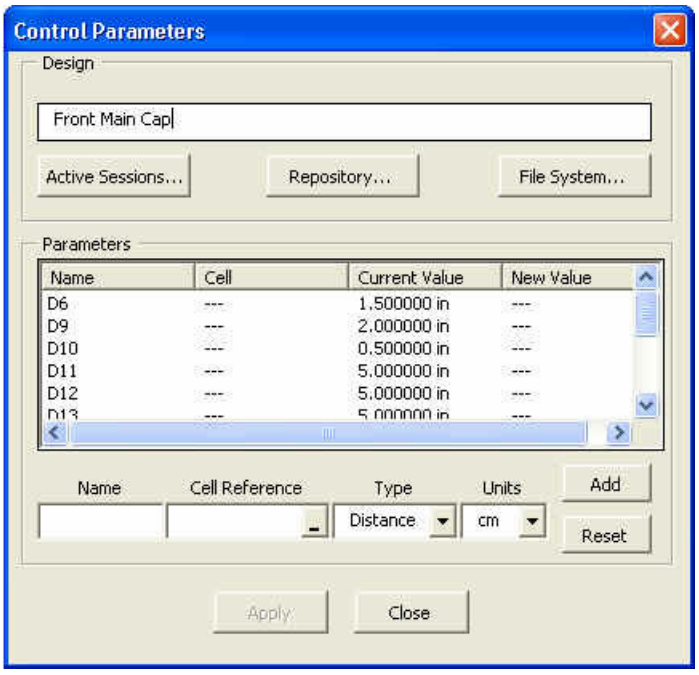
2. Open the part that is linked to the spreadsheet.

3. Launch Microsoft Excel.

4. Open the spreadsheet of prepared parameters.

5. From the **Tools** menu in Excel, select **Alibre Design Add-In > Control Parameters**. (For **Excel 2007**, select the **Add-Ins** tab. Choose **Alibre Design Add-In > Control Parameters**.) The Control Parameters dialog appears. (You may be prompted to save the workbook, the Excel file, before using it to control parameters.) The name of the open part is displayed in the Design field and its dimensions appear in the Parameters table.

TIP: If several parts are open in Alibre Design, click **Active Sessions** in the Control Parameters dialog. The Active Sessions dialog appears. Select the desired part and click **OK**.



6. In **Parameters**, click one of the parameters. Its name appears in the Name field.

7. For **Cell Reference**, click the field; then, click the cell on the Excel spreadsheet that the parameter should reference.

   Or

   Click the button. The cell reference box appears. Click the desired spreadsheet cell; then click the button to return to the Control Parameters dialog.

Note: If the spreadsheet is moved, or any folders are renamed, the link from the cell to the part dimension will be broken. To reestablish the link, use the Equation Editor dialog to *re-link the spreadsheet* (see "Re-linking a Spreadsheet to a Part" on page 173) to the part.

8. In **Type**, select the parameter type: distance, angle, or count.

9. In **Units**, set the desired units of measurement (for distance and angle parameters).

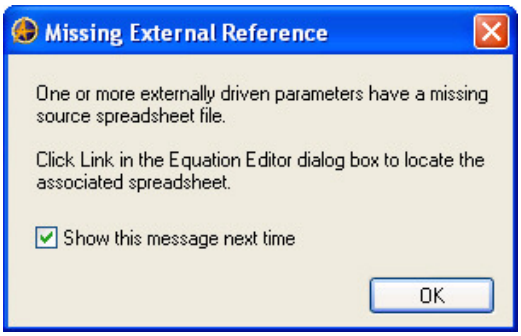
10. Click **Modify**.

11. To add a new parameter, click **Reset**. The entry fields are cleared. In **Name**, type a name for the new parameter; then follow steps 6-10 above.

12. Click **Close**. The open part in Alibre Design is modified to the new parameters.

9.9.4 Re-linking a Spreadsheet to a Part

If a spreadsheet file used in driving a design is moved, or folders are renamed in the path to their locations, their links to the parts they drive will be broken. The next time you open the Equation Editor dialog, this warning will appear.


Missing External Reference
One or more externally driven parameters have a missing source spreadsheet file.
Click Link in the Equation Editor dialog box to locate the associated spreadsheet.
Show this message next time
OK

To reestablish the links between spreadsheets and parts, use the link button in the Equation Editor dialog.

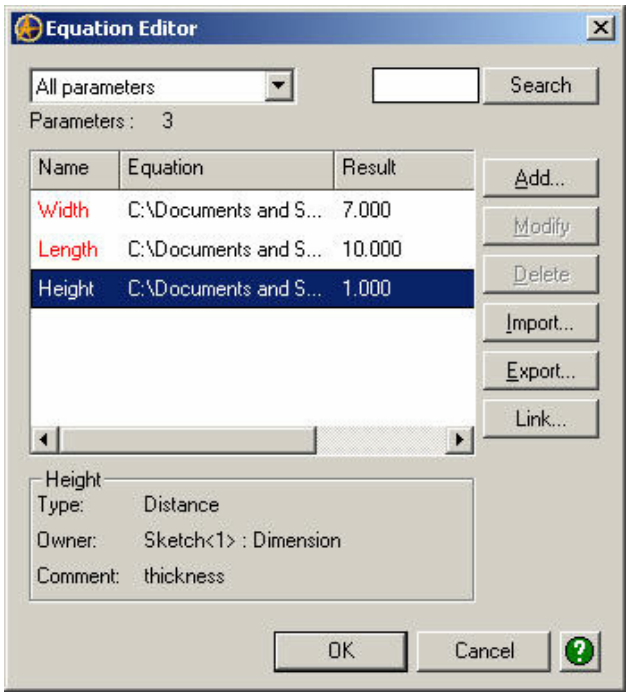
➢ ***To reestablish the link between a part and a spreadsheet***

1. With the affected part open, click the **Equation Editor** tool.

   Or

   From the **Tools** menu, select **Equation Editor**. The Missing External Reference dialog appears.

2. Click **OK**. The Equation Editor dialog appears, with the affected parameters displayed in red.



3. Click an affected parameter and click **Link**. The Link Parameter dialog appears.

4. Click **Windows Folders,** then **Browse,** if the associated spreadsheet is in a Windows folder. The Browse for Spreadsheet File dialog opens.

5. In **Look In**, browse to the location of the spreadsheet.

6. Click the spreadsheet's file name to select it.

7. Click **Open**. The new file path to the spreadsheet appears in the browse field.

8. Click **Apply to all such parameter**s to apply the changes to all parameters driven by the same spreadsheet.

9. Click **OK**.

10. Repeat steps 4-10 if any affected parameters are driven by a separate spreadsheet.

9.10 3D PDF Publishing

Alibre Design supports publishing parts, assemblies, drawings, and BOMs to PDF file format. Once published, these files can be viewed with the Adobe Reader.

There are two levels of PDF publishing:

- Model Only Publishing
- Full Publisher Module

Note: Users with licenses for Alibre Design Xpress and Standard have access to Model Only Publishing. Users with licenses for Alibre Design Professional and Expert have access to Full Publishing. PDF Publishing is not supported for licenses of Alibre Design Basic.

Model Only Publishing Features

- Ability to publish PDF files with interactive 3D model, including rotate, pan, and zoom.
- PDF file can be viewed with Adobe Reader 7.0.8 and above.

Full Publisher Module Features

- Ability to publish PDF files with:
  - Interactive 3D model, including rotate, pan, and zoom
  - Assembly tree
  - Drawings
  - BOMs
  - Exploded view animation with ordered steps
  - Saved views
- PDF file can be viewed with Adobe Reader 7.0.8 and above.

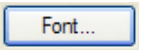
9.10.1 Creating a PDF File

➢ ***To create a PDF file of a part, sheet metal part, or assembly:***

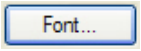
1. From the **File** menu, select **Publish to PDF**, or select the **Publish to Acrobat PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard opens, and the 3D Model tab is active.

2. Select a template to use. (See *PDF Publishing Templates* (on page 179) for more information)

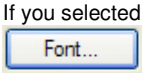
3. Click **Next**.

4. If you want to add a header, select the **Title** tab and enter a title. Click the Font button  to change the font of the Title.

   Tip: The Title can be one line only, and the text will not wrap.

5. If you want to add a footer, select the **Footer** tab and enter a footer. Click the Font button  to change the font of the Footer.

   Tip: The Footer can be one line only, and the text will not wrap.

6. If you selected a template that includes text, select the **Body Text** tab(s) and enter text as necessary. Click the Font button  to change the font of the text. To include a carriage return, press CTRL + Enter on your keyboard.

7. Click **Next**.

8. Check the **Views** you want to include in your PDF, if any. The default view (the state the model was in at the time you began the publishing process) is always included.

9. Click **Next**.

10. In the **Page Layout** section, select the page layout to which you want to publish.

11. Check the **Put my logo on the top right corner** option to include an image in the PDF; then, click Browse to find and select the image.

12. In **Save Location**, browse to the location where you want to save the PDF file.

    Note: If you choose to save to an existing file, after clicking Publish, you will be prompted to choose if you want to overwrite the existing file or add the published page to the existing file at the end as an additional page.

13. Choose **Preview** if you would like to view the PDF file before saving it. This creates a temporary file that is deleted if you do not publish the file.

14. Click **Publish**.

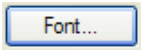
The PDF file will be created and saved to the location you specified, and can be viewed with Adobe Reader 7.0.8 or higher.

➢ ***To create a PDF file of a section view:***

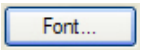
1. You must be viewing the section view in the work area.
2. From the **File** menu, select **Publish to PDF**. The PDF Publishing Wizard opens, and the 3D Model tab is active.
3. Continue following the steps listed above for creating a PDF file of a part, assembly, or sheet metal part.

➢ ***To create a PDF file of an assembly in an exploded view:***

Note: You must have a license for full publishing. You must be viewing an assembly in the exploded state.

1. From the **File** menu, select **Publish to PDF**, or select the **Publish to Acrobat PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard opens, and the 3D Model tab is active.
2. Select a template to use. (See *PDF Publishing Templates* (on page 179) for more information)
3. Click **Next**.
4. If you chose one of the templates that includes exploded view steps:
   a. In **Publish State**, choose to publish in Exploded or Unexploded State. This will determine which state the model is displayed in the PDF.
   b. Check the Include Explosion steps and descriptions box if you would like to include the steps in the PDF. Click the Font button if you would like to change the font of the exploded view steps in the PDF. The text "PDF Publishing" will update to reflect the new font.
   c. Click **Next**.
5. If you want to add a header, select the **Title** tab and enter a title. Click the Font button  to change the font of the Title.

Tip: The Title can be one line only, and the text will not wrap.

6. If you want to add a footer, select the **Footer** tab and enter a footer. Click the Font button  to change the font of the Footer.

Tip: The Footer can be one line only, and the text will not wrap.

7. Click **Next**.
8. Check the **Views** you want to include in your PDF, if any. The default view (the state the model was in at the time you began the publishing process) is always included.
9. Click **Next**.
10. In the **Page Layout** section, select the page layout to which you want to publish.
11. Check the **Put my logo on the top right corner** option to include an image in the PDF; then, click Browse to find and select the image.
12. In **Save Location**, browse to the location where you want to save the PDF file.

Note: If you choose to save to an existing file, after clicking Publish, you will be prompted to choose if you want to overwrite the existing file or add the published page to the existing file at the end as an additional page.

13. Choose **Preview** if you would like to view the PDF file before saving it. This creates a temporary file that is deleted if you do not publish the file.

14. Click **Publish**.

The PDF file will be created and saved to the location you specified.

➢ ***To create a PDF file of images or text:***

1. From the **File** menu, select **Publish to PDF**, or select the **Publish to Acrobat PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard opens.

2. Select the **General** tab.

3. Select a template to use. (See *PDF Publishing Templates* (on page 179) for more information)

4. Click **Next**.

5. If you selected a template that includes an image, click **Browse** to find and select the image you want to publish.

6. Depending on which template you selected, select the Title, Footer, and/or Body Text tabs and enter the text you want to appear in each location.

Tip: The Title and Footer can be one line only, and the text will not wrap. To include a carriage return in the Body Text section, press CTRL + Enter on your keyboard.

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7. Click **Next**.

8. In the **Page Layout** section, select the page layout to which you want to publish.

9. If you chose the Image with Text template or the Text Only template, check the **Put my logo on the top right corner** to include an image in the PDF; then, click Browse to find and select the image.

10. In **Save Location**, browse to the location where you want to save the PDF file.

Note: If you choose to save to an existing file, after clicking Publish, you will be prompted to choose if you want to overwrite the existing file or add the published page to the existing file at the end as an additional page.

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11. Choose **Preview** if you would like to view the PDF file before saving it. This creates a temporary file that is deleted if you do not publish the file.

12. Click **Publish**.

The PDF file will be created and saved to the location you specified.

➢ ***To create a PDF file of a 2D drawing:***

Notes:

You must have installed PDFCreator during the installation of Alibre Design.

You must have Full Publishing capability.

PDFCreator cannot be installed on the Vista operating system.

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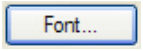
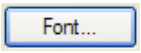
1. From the **File** menu, select **Publish to PDF**, or select the **Publish to Acrobat PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard opens, and the BOM/Drawing tab is active. The Drawing Template option is selected.

2. Click **Next**.

3. If you want to add a header, select the **Title** tab and enter a title. Click the Font button 

to change the font of the Title.

Note: The Title can be one line only, and the text will not wrap.

4. If you want to add a footer, select the **Footer** tab and enter a footer. Click the Font button 

to change the font of the Footer.

Note: The Footer can be one line only, and the text will not wrap.

5. Click **Next**.
6. In the **Page Layout** section, select the page layout to which you want to publish.
7. Check the **Put my logo on the top right corner** to include an image in the PDF; then, click Browse to find and select the image.
8. In **Save Location**, browse to the location where you want to save the PDF file.

Note: If you choose to save to an existing file, after clicking Publish, you will be prompted to choose if you want to overwrite the existing file or add the published page to the existing file at the end as an additional page.

9. Choose **Preview** if you would like to view the PDF file before saving it. This creates a temporary file that is deleted if you do not publish the file.
10. Click **Publish**.

The PDF file will be created and saved to the location you specified, and can be viewed Adobe Reader 7.0.8 or higher.

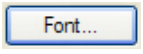
➢ ***To create a PDF file of a BOM:***

Notes:

You must have installed PDFCreator during the installation of Alibre Design.

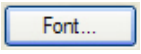
You must have Full Publishing capability.

PDFCreator cannot be installed on the Vista operating system.

1. From the **File** menu, select **Publish to PDF**, or select the **Publish to Acrobat PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard opens, and the BOM/Drawing tab is active. The BOM Template option is selected.
2. Click **Next**.
3. If you want to add a header, select the **Title** tab and enter a title. Click the Font button 

to change the font of the Title.

Tip: The Title can be one line only, and the text will not wrap.

4. If you want to add a footer, select the **Footer** tab and enter a footer. Click the Font button 

to change the font of the Footer.

Tip: The Footer can be one line only, and the text will not wrap.

5. Click **Next**.
6. In the **Page Layout** section, select the page layout to which you want to publish.
7. Check the **Put my logo on the top right corner** to include an image in the PDF; then, click Browse to find and select the image.
8. In **Save Location**, browse to the location where you want to save the PDF file.

Note: If you choose to save to an existing file, after clicking Publish, you will be prompted to choose if you want to overwrite the existing file or add the published page to the existing file at the end as an additional page.

9. Choose **Preview** if you would like to view the PDF file before saving it. This creates a temporary file that is deleted if you do not publish the file.

10. Click **Publish**.

The PDF file will be created and saved to the location you specified, and can be viewed Adobe Reader 7.0.8 or higher.

9.10.2 Continuing with an Existing PDF

If you have previously created a PDF file, you can continue adding data to that PDF at a later time.

➢ ***To continue working on an existing PDF:***

1. Open the file containing the model, drawing, or BOM that you want to include in the PDF.

2. From the **File** menu, select **Continue Previously Created PDF**, or select the **Continue Previously Created PDF** tool from the PDF Publishing toolbar. The PDF Publishing Wizard dialog appears.

3. Select the appropriate radio button to choose whether you want to **Append to the last PDF you created**, or **Append to a different PDF**.

4. If you chose to append to a different PDF, click **Browse** to choose the PDF.

5. Click **Next**.

6. Continue in the PDF Publishing wizard as described in *Creating a PDF File* (on page 175).

9.10.3 PDF Publishing Templates

Templates available when creating a PDF from a Part, Sheet metal Part, or Assembly Workspace:

	Model Only, Full Page	The published PDF file will have one page which shows the 3D model.
	Model Only, Half Page	The published PDF file will have one page which shows the 3D model only in the upper half of the PDF page. The rest of the PDF page will be blank.
	Model with text in lower half	The published PDF file will have one page which shows the 3D model in the upper half of the PDF page. The lower half of the PDF page will include your custom text.

Templates available when creating a PDF from an Assembly Workspace while in an exploded view (only available if you have a license for full publishing):

Exploded View, Full Page — The published PDF file will have one page which shows the assembly in its exploded view and a title (if one is specified).

Exploded view, Half Page — The published PDF file will have one page which shows the assembly in its exploded view and a title (if one is specified) only in the upper half of the PDF page. The rest of the PDF page will be blank.

Exploded view, Top, with Exploded View Step Information — The published PDF file will show the assembly in the exploded view in the upper half of the PDF page with all of the steps listed under it.

Exploded view, Left, with Exploded View Step Information — The published PDF file will show the assembly in the exploded view on the left side of the PDF page with all of the steps listed on the right.

Templates available when creating a PDF of an image or of text only:

Cover Page — The published PDF file will have one page which shows the image in the top half, and your custom text in the bottom half.

Image with Text — The published PDF file will have one page which shows the image in the top half, and your custom text in the bottom half, along with any specified Title and Footer.

Text Only — The published PDF file will include your custom text.

Templates available when creating a PDF from a Drawing Workspace:

Drawing — The published PDF file will have one page which shows the sheet that was active when the publishing process was started.

Templates available when creating a PDF from a BOM Workspace:

Bill of Material	The published PDF file will have one page which shows Bill of Material.

9.10.4 Viewing Published PDF Files

After you have published a PDF file, you can view the PDF using the Adobe Reader version 7.0.8 or higher.

➢ ***To view a published PDF file:***

1. Ensure that you have installed Adobe Reader.
2. Browse to the location where you saved the file.
3. Double-click the file, and it will open in the Adobe Reader.

➢ ***To view any additional views included in the file:***

Not all files will include additional views. The views must have been selected during the publishing process.

1. Ensure that you have installed Adobe Reader. (Must be version 7.0.8 or higher)
2. Browse to the location where you saved the file.
3. Double-click the file, and it will open in the Adobe Reader.
4. Left-click once on the image of the model,

   Or,

   Right-click the image of the model and choose **Enable 3D**.
5. From the Views drop down list, select the view you want.

Note: You can also select the Model Tree tab to see a list of all of the published views.

➢ ***To view the explosion animation of an assembly***

Not all files will have the explode animation. The file must have been published that way originally. This capability is available with the Full Publishing package.

1. Ensure that you have installed Adobe Reader. (Must be version 7.0.8 or higher)
2. Browse to the location where you saved the file.
3. Double-click the file, and it will open in the Adobe Reader.
4. Left-click once on the image of the model,

   Or,

   Right-click the image of the model and choose Enable 3D.
5. Click on any of the steps in the list to see the assembly explosion at that step.
6. Click Reset, then Explode to view the animation of the entire explosion.
7. To view the implosion, click Reset, and then Implode.

Note: You must click Reset each time you want to select either Explode or Implode for the animation to work correctly.

9.10.5 Improving the Quality of Published PDF Files

Following these suggestions can improve the quality of your published PDF files. Before beginning the publishing process:

- Make sure your workspace window is maximized
- Choose a white background for your work area
- Make the model fill as much of the work area as possible

9.10.6 Publishing a Model to HTML

You can send the image of a model directly to an HTML file, if necessary.

➢ ***To publish the model to an HTML file:***

1. Open the file containing the model you want to publish.
2. From the File menu, select **Publish 3D PDF of model only in HTML**, or select the **Publish 3D Model in HTML** tool from the PDF Publishing toolbar. The PDF Publishing Wizard dialog appears.
3. In **Save Location**, type the location where you want to save the file, or click the Browse button to specify the location.
4. In **File Names**, enter a name for both the PDF file and the HTML file.

   You must provide a filename for both PDF and HTML.The HTML file contains a link to the PDF, so both files are required for the HTML file to open correctly.

5. Click **Publish**.

9.10.7 Printing 3D Models

You can print the images from your 3D workspace. The print function will print all that is displayed, so you may want to hide planes, axes, and other items.

➢ ***To print:***

1. From the File menu, select **Print**.
2. If you want to print using a white background, check the option **Use white background**.
3. Choose the printer and the number of copies, and click **OK**.

9.10.8 Annotations

You can insert 3D annotations in part and assembly workspaces. The following annotation types are supported in 3D workspaces:

- Notes
- Datums
- Datum Targets
- Feature Control Frames
- Surface Finishes
- Weld Symbols
- The methods used to insert annotations for drawing and model workspaces are the same. Refer to *Inserting an Exploded View* (on page 223) for detailed information related to inserting annotations.

9.10.9 Troubleshooting Failed Features

A feature can fail to generate properly due to numerous reasons. If a feature fails to generate correctly, a message is generated after the initial creation, and a red X is displayed on the design icon in the Design Explorer, as illustrated below.

Common causes of feature generation failure include:

- The sketch used to define the feature profile contains open or overlapping figures (e.g. an extrude boss feature fails if the profile sketch contains any open ends).
- Improperly specifying the feature parameters (e.g. interchanging the path sketch and profile sketch involved in a sweep boss).
- Modifying an upstream feature causes a downstream feature to fail.

➢ ***To troubleshoot a failed feature:***

- Right-click the feature and select **Status** from the pop-up menu. A dialog appears containing information related to the cause of the feature failure.
- Right-click the feature and select **Edit**. The feature properties dialog appears. Modify the conditions, if applicable.
- If the failure is a result of an incorrect sketch, edit the sketch and correct accordingly.

9.10.10 Viewing Constituents

You can view the constituents of Alibre Design Parts, Sheet metal Parts, Assemblies, Drawings, and Bills of Material.

➢ ***To view constituents in the Windows File System:***

1. In the Alibre Design Home Window, from the **Tools** menu, select **Show Constituents**. The **Show Constituents** dialog appears.
2. Browse to locate the item in the dialog.
3. Click **Open**. The **Constituents** dialog appears, showing all items that are related to the selected item.

   Or

4. In the Windows File System, right-click an Alibre Design file and select **Constituents**. The **Constituents** dialog appears, showing all items that are related to the selected item.

9.10.11 Display Optimization

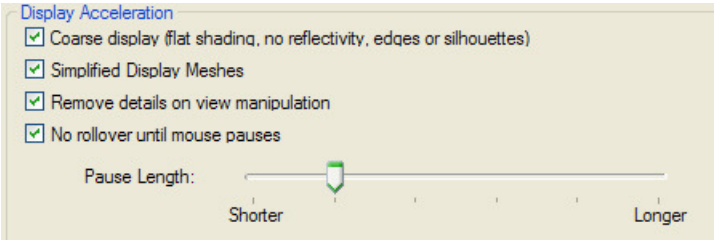
Display Acceleration

You can speed up the processing of large designs by entering Display Acceleration mode. This is achieved by simplifying the design according to user-defined options. This option can be used in designs (including parts, assemblies, sheet metal parts, exploded assemblies, and Design Booleans), but not in drawings or BOMs. Some menu items will be unavailable while in Display Acceleration.

➢ ***To set preferences for Display Acceleration:***

1. In a part or assembly workspace, from the **File** menu, select **Properties**.

2. Select the **Display** tab.



3. Check your desired options for Display Acceleration:

   - **Coarse Display** - Flat shading will be selected, no reflectivity will be used, visible and silhouette edges will not be shown. If this option is checked, changing the display options (wireframe, shaded, etc.) will not change the display of the model.
   - **Simplified Display Meshes** - The complexity of meshes will be reduced, which will reduce the visual precision of the parts.
   - **Remove Details on View Manipulation** - During rotate, pan, and zoom operations, some faces and small parts will be left out of the display. These will be returned to the display after the operation is completed.
   - **No rollover until mouse pauses** - If this option is checked, you will not see any items highlight as you move the cursor until the mouse has paused for the designated amount of time. At that time, the item where the cursor is paused will highlight.

4. Choose **Apply**, then **Close**.

➢ ***To enter Display Acceleration:***

From the **View** menu, select **Display Acceleration.**

Or

Select the **Display Acceleration** tool from the View Toolbar.

➢ ***To exit Display Acceleration:***

From the **View** menu, select Display Acceleration.

Or

Select the **Display Acceleration** tool from the View Toolbar.

Note: If you save a model with Display Acceleration active, it will remain in Display Acceleration when it is opened again.

CHAPTER 10

Design Configurations

Design configurations allow you to create multiple variations of a part, sheet metal part, or assembly and maintain them in a single workspace. When a part with configurations is included in an assembly, you can specify which configuration you want to see in the assembly. In addition, when you are creating a drawing, you can specify which configuration of a part or assembly will be inserted into a drawing view.

The ability to create and edit Design Configurations is available in Alibre Design Professional and higher versions. However, every part, sheet metal part, and assembly workspace will contain one configuration when it is opened. This is true regardless of which version of Alibre Design you are using. In versions that do not include the ability to create or edit configurations, this will be the only configuration.

For example, if you open a design model in Alibre Design Basic that was originally modeled in Professional, and has multiple configurations, you will see the configurations listed in the Design Explorer, but you will only be able to edit the configuration that was active when the model was last saved.

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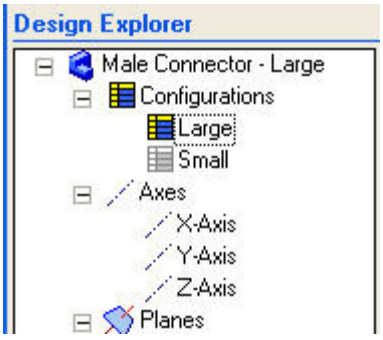
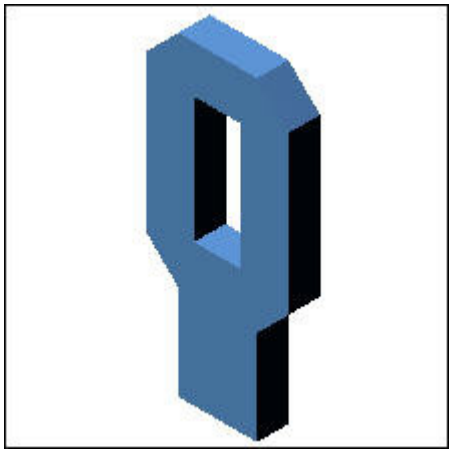
Design Configurations Overview 185
Creating Part and Sheet Metal Part Configurations 188
Assembly Configurations 190
Using Configurations in Drawings 199
Using Configurations in a BOM 200
Using the Equation Editor with Configurations 202

10.1 Design Configurations Overview

Below are illustrations for the use of Design Configurations in a part model named "Male Connector":

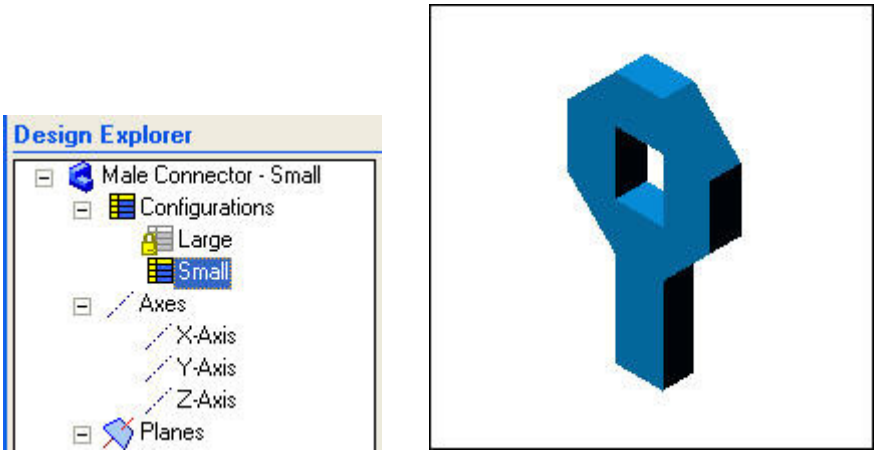
Configuration of the Large Version of a Male Connector





In the Design Explorer, the part model name has "- Large" appended to it, noting the active configuration. The configuration called "Large" has the active icon next to it (see *Lock Properties Icon Descriptions* (see "Lock properties represented in the Design Explorer" on page 186) for information about each icon). The configuration called "Small" has the inactive, unlocked icon. The model in the work area is displayed as shown.

Configuration of the Small Version of a Male Connector



In the Design Explorer, the part model name now has "- Small" appended to it, noting the new active configuration. The configuration called "Large" has the inactive, locked icon next to it. The configuration called "Small" has the active icon. The model in the work area is displayed as shown.

10.1.1 Lock properties represented in the Design Explorer

Each configuration of a part, sheet metal part, or assembly listed in the Design Explorer has an icon that represents the lock properties of that configuration. The table below describes each icon.

Icon	Lock State	Description
	Active	All changes made apply to this active configuration. Only one configuration can be active at a time.
	Unlocked	All of the changes made to the active configuration will be applied to unlocked configurations.
	Locked	None of the changes made to the active configuration that are controlled by configuration lock properties will be applied to these locked configurations.
	Partially Locked	Some of the changes made to the active configuration will apply to these partially locked configurations, depending on the particular locks that are set for each one.
	Missing	The configuration that was set for this assembly constituent is missing, usually because it has been deleted. To resolve this, activate one of the other existing configurations by double-clicking it.

10.1.2 Editing Properties of Configurations

You can edit the lock properties of configurations in a design to control how each of the configurations update when you are making changes to the active configuration.

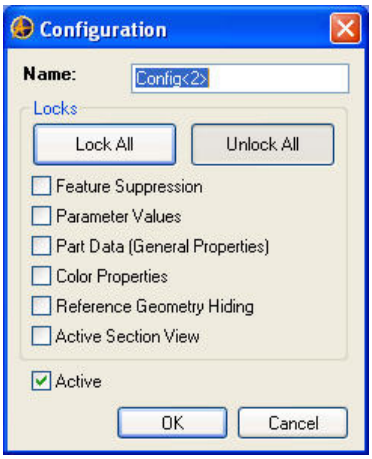
➢ ***To edit the lock properties of a configuration:***

1. From the **Edit** menu, select **Edit Config<2>**. (The configuration that is available to edit from this menu will be the one you have highlighted in the Design Explorer.)

   Or

   Right-click the configuration in the Design Explorer and select **Edit**.

   The Configuration dialog appears. This dialog is similar to the New Configuration dialog.



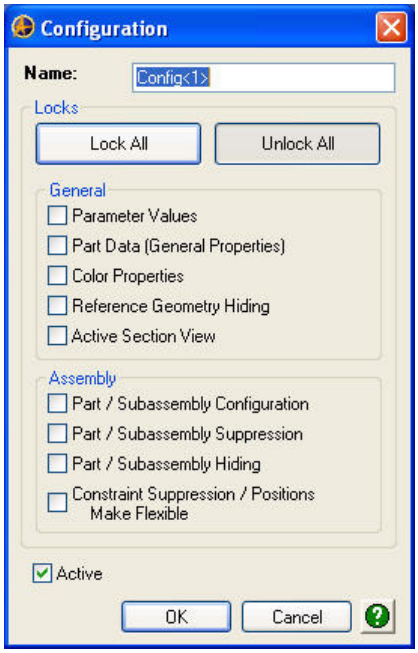


Figure 77: Edit Part Configuration Dialog

2. Check the options you want to Lock for this configuration.

3. Select **OK** to accept the selections.

To make a configuration active Double-click the configuration you want to activate; Or

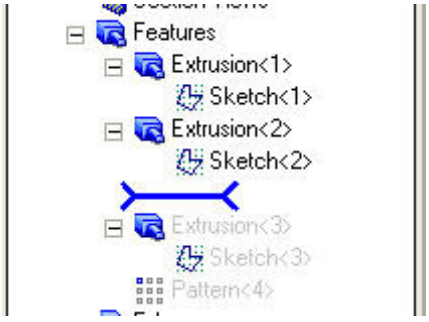
Right-click the configuration you want to activate and select **Activate**;

Or

Follow steps 1 and 2 above, and check the box next to the option **Active**, then select **OK**.

10.1.3 Regeneration of Design Configurations

When Alibre Design models are regenerated, each feature in the Design Explorer is calculated in the order it appears. All features that appear above the blue "dog bone" are features that have been regenerated. Features below the blue "dog bone," as well as suppressed features, are not generated.



When modeling with multiple configurations, it is important to realize that as features are created in the active configuration, the features for inactive configurations are recorded but not executed. This means that inactive configurations are not updated while you are editing the active configuration.

Using the "Update Activated Configuration" option, you can choose if you want Alibre Design to automatically compute all newly added features when you switch a configuration from inactive to active.

➢ ***To set the Update Activated Configuration option:***

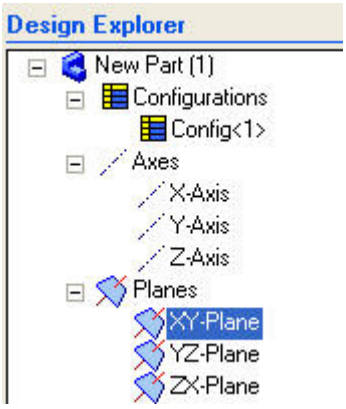
In a Part Workspace, from the **Features** menu, choose **Update Activated Configuration**. When this option is checked, it is activated. (This is the default setting.) When it is unchecked, it is turned off.

- If Update Activated Configuration is **checked**: When you activate a configuration, it will automatically be computed up to the dog bone position in the configuration you just edited.
- If Update Activated Configuration is **unchecked**: When you activate a configuration, no new features will be computed. The dog bone is located below the last valid feature in the newly activated configuration. This is useful if you want to modify data that does not require feature regeneration, such as Property Data.

Note: When this option is off, the position of the dog bone can vary greatly between each configuration.

10.2 Creating Part and Sheet Metal Part Configurations

When a new part or sheet metal part workspace is opened, the Configurations category will appear in the Design Explorer as the first category in the list.



Under the Configurations category, Config <1> is listed. Every part contains at least one configuration. This is true for all versions of Alibre Design.

➢ ***To create a new configuration:***

Note: Not all versions of Alibre Design include the ability to create new configurations.

1. From the **Insert** menu, select **Configuration**; Or right-click on **Configurations** in the Design Explorer and choose **New Configuration**; Or select the **Configuration** tool from the Inspection toolbar. The New Configuration dialog appears.

2. In the **Name** field, enter a name for the configuration.

3. In the **Copy From** drop down menu, choose which configuration to use to start the new one. The **Locks** field is populated with the properties from the configuration you choose. You can modify these as needed.

4. In the **Locks** field, set the lock requirements for the configuration.

   a. **Lock All** will check all of the properties

   b. **Unlock All** will uncheck all of the properties

   The following table describes how each lock function will affect the model behavior:

Individual Lock	Locked Behavior (Checked)	Unlocked Behavior (Unchecked)
Feature suppression	Any features added to the active configuration will be added as suppressed features in each locked configuration. Changes to suppression state (suppress/unsuppress) for already existing features in the active configuration will be ignored in locked configurations.	Any features added to the active configuration will also be added to each unlocked configuration. Changes to suppression state of already existing features in the active configuration will also be applied to each unlocked configuration.
Parameter values	Any parameters added to the active configuration will also be applied with their initial values to all locked configurations. However, any changes to parameter values will not be applied to locked configurations.	Any parameters added to the active configuration will be applied with their initial values to all unlocked configurations. In addition, any changes to parameter values will also be applied to all unlocked configurations.
Part Data (General Properties)	Changes to the active configuration's Part Data property values will be ignored in all locked configurations. Note that the Name, Number, and Description fields in the Part Data will be the same for all configurations. The locks apply to the property values.	Changes to the active configuration's Part Data property values will be applied to all unlocked configurations.

Individual Lock	Locked Behavior (Checked)	Unlocked Behavior (Unchecked)
Color Properties	Changes to the edge and/or face colors of the active configuration will be ignored in all locked configurations.	Changes to the edge and/or face colors of the active configuration will also be applied to all unlocked configurations.
Reference Geometry Hiding	Hiding and unhiding of reference geometry in the active configuration is ignored in all locked configurations.	Hiding and unhiding of reference geometry in the active configuration will be applied to all unlocked configurations.
Active Section View	Choosing to activate or deactivate the view of a 3D section view in the active configuration is ignored in all locked configurations.	Activating or deactivating the view of a 3D section view in the active configuration also changes the view state of all unlocked configurations.

5. Check the **Active** checkbox to make the new configuration the active one.

6. Select **OK** to create the new configuration. The new configuration will appear in the Design Explorer. In addition, once a second configuration has been created, the part name in the Design Explorer and in the workspace title bar will change to read "Part Name – Active Configuration Name."

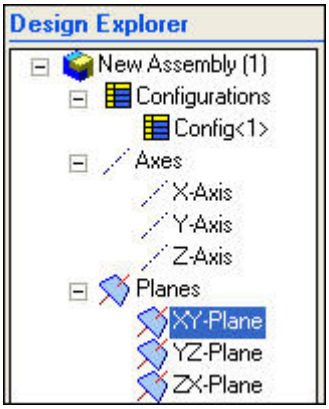
Note: If you have only a single configuration, the part name in the Design Explorer and in the workspace title bar will read only "Part Name." There will be no configuration name appended to the name.

10.2.1 Helpful Notes on Design Configurations in Parts

- Moving or resizing a sketch figure manually (such as by clicking and dragging) applies to all configurations. Moving or resizing a sketch figure using dimensions does not necessarily apply to all configurations. (Dimension values are parameters that can be different in each configuration.)
- All configurations have the same set of dimensions. A dimension added to one configuration will be added to all configurations. Values of dimensions are parameters that can differ between configurations.
- All configurations have a common set of available features. A feature deleted from one configuration is deleted from all configurations. (Features can be suppressed in individual configurations.)
- All configurations have a common feature order. Features reordered in one configuration are reordered in all configurations.
- In the Design Explorer, configurations are listed in the order they were created. You can reorder them, if you are aware that relationships between features that are broken as a result of reordering will cause the affected features to fail.
- You cannot delete the active configuration. If only one configuration exists, it is by default the active configuration, and the **Delete** option will be disabled.

10.3 Assembly Configurations

As with part workspaces, when a new assembly workspace is opened, the Configurations category will show up in the Design Explorer as the first category in the list.

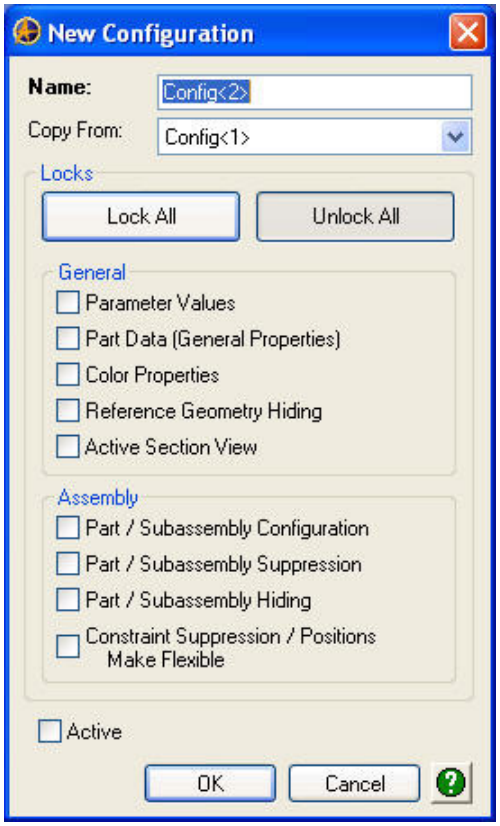


Under the **Configurations** category, Config<1> is listed. Every assembly contains at least one configuration. This is true for all versions of Alibre Design.

➢ ***To create a new configuration:***

Note: Not all versions of Alibre Design include the ability to create new configurations.

1. From the **Insert** menu, select **Configuration**; Or right-click on **Configurations** in the Design Explorer and choose **New Configuration**; Or select the Configuration tool from the Inspection toolbar. The New Configuration dialog appears.



2. In the **Name** field, enter a name for the configuration.

3. In the **Copy From** drop down menu, choose which configuration to use to start the new one. The **Lock** properties from the configuration you choose will populate in the **Locks** field. You can modify these as needed.

4. In the **Locks** field, set the lock requirements for the configuration.

   a. **Lock All** will check all of the properties.

   b. **Unlock All** will uncheck all of the properties.

The following table describes how each lock function will affect the model behavior:

Individual Lock	Locked Behavior (Checked)	Unlocked Behavior (Unchecked)
General		
Parameter values	Any parameters added to the active configuration will also be applied with their initial values to all locked configurations. However, any changes to parameter values will not be applied to locked configurations.	Any parameters added to the active configuration will be applied with their initial values to all unlocked configurations. In addition, any changes to parameter values will also be applied to all unlocked configurations.
Part Data (General Properties)	Changes to the active configuration's Part Data property values will be ignored in all locked configurations. Note that the Name, Number, and Description fields in the Part Data will be the same for all configurations. The locks apply to the property values.	Changes to the active configuration's Part Data property values will be applied to all unlocked configurations.
Color Properties	Changes to the edge and/or face colors of the active configuration will be ignored in all locked configurations.	Changes to the edge and/or face colors of the active configuration will also be applied to all unlocked configurations.
Reference Geometry Hiding	Hiding and unhiding of reference geometry in the active configuration is ignored in all locked configurations.	Hiding and unhiding of reference geometry in the active configuration will be applied to all unlocked configurations.
Active Section View	Choosing to activate or deactivate the view of a 3D section view in the active configuration is ignored in all locked configurations.	Activating or deactivating the view of a 3D section view in the active configuration also changes the view state of all unlocked configurations.
Assembly		
Part / Subassembly Configuration	Changing the configuration of a component that is used in the active assembly configuration does not affect any locked assembly configurations.	Changing the configuration of a component that is used in the active assembly configuration will also be changed in all unlocked assembly configurations.
Part / Subassembly Suppression	Components that are inserted in the active assembly configuration will be inserted as suppressed components in all locked assembly configurations. For already existing components, changing between suppressed and unsuppressed will not affect their suppression states in any locked assembly configurations.	Components that are inserted in the active assembly configuration will also be inserted in all unlocked assembly configurations (in the same state). For already existing components, changing between suppressed and unsuppressed will change their suppression states in all unlocked assembly configurations.

Individual Lock	Locked Behavior (Checked)	Unlocked Behavior (Unchecked)
Part / Subassembly Hiding	Hiding and unhiding of assembly components in the active assembly configuration is ignored in all locked assembly configurations.	Hiding and unhiding of assembly components in the active assembly configuration also applies to all unlocked configurations.
Constraint Suppression / Positions / Make Flexible	Newly added constraints are suppressed. Modifying the suppression state of an existing constraint has no effect. Changes to the make- flexible state are ignored. The position of unconstrained constituents is locked.	Newly added constraints are also applied to unlocked configurations. Modifying the suppression state of an existing constraint is applied to unlocked configurations. Changes to the make- flexible state are applied to all unlocked assembly configurations. If the position of a component in the active configuration is modified, the position of that component in all unlocked assembly configurations will change to match.

5. Check the **Active** checkbox to make the new configuration the active one.

6. Select **OK** to create the new configuration. The new configuration will appear in the Design Explorer. In addition, once a second configuration has been created, the assembly name in the Design Explorer and in the workspace title bar will change to read "Assembly Name – Active Configuration Name."

Note: If you have only a single configuration, the assembly name in the Design Explorer and in the workspace title bar will read only "Assembly Name." There will be no configuration name appended to the name.

10.3.1 Inserting Configurations of Parts or Subassemblies

When you insert a part or subassembly into a higher level assembly workspace, the active configuration of that model will automatically be inserted. After it has been inserted, you can change the configuration of the part or subassembly that is used.

➢ ***To change the configuration that is used in an assembly***

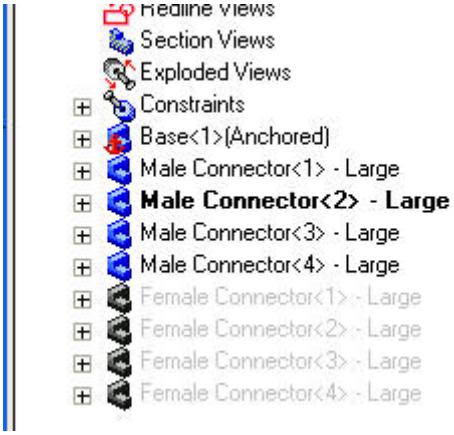
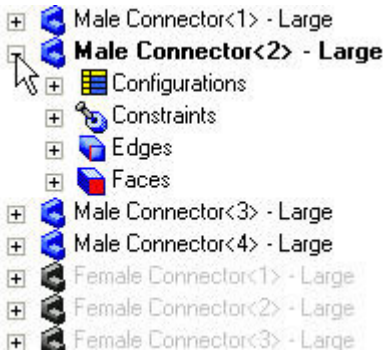
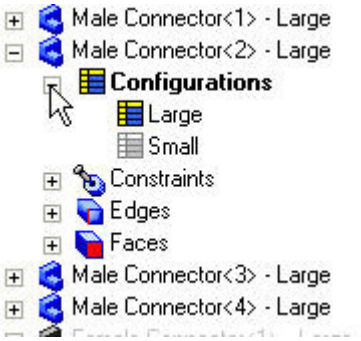


Figure 78: Design Explorer Showing a List of Parts in an Assembly

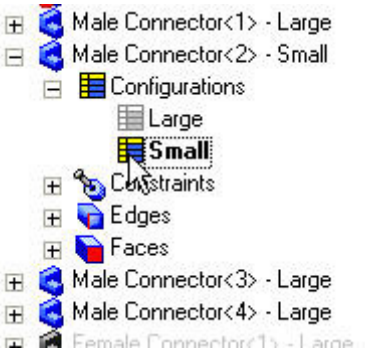
1. Find the part or subassembly that you want to set a new configuration for, and expand it to see each of the categories for that component.



2. Expand the Configurations category. In this example, the configuration called "Large" is currently active.



3. Double-click the configuration you want to make active in this assembly, or, right-click the configuration and select **Activate**. In this example, I double-clicked the configuration called "Small".



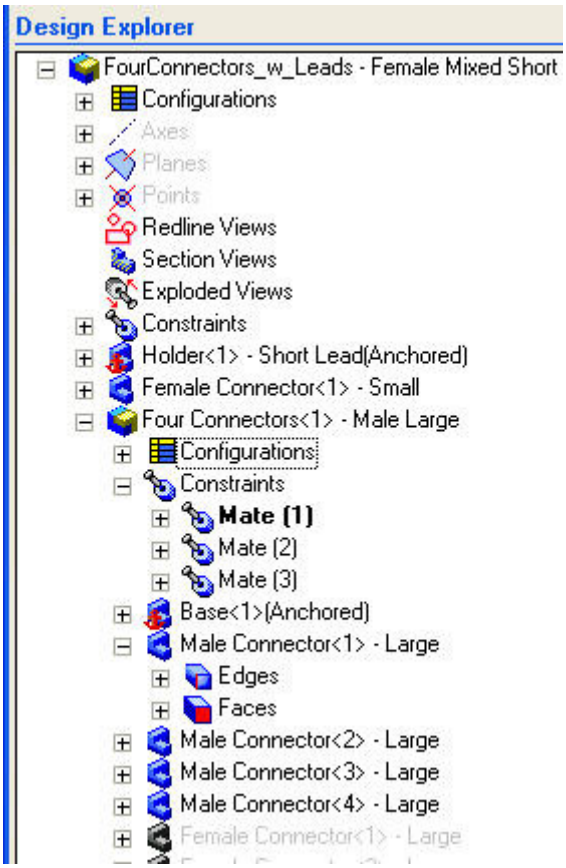
The new configuration will become active in this assembly, and will be shown in the work area. This will not affect which configuration is active in any other assembly, or in the original model workspace.

Note: Configuration information is only displayed for immediate children of the root assembly. Parts nested within sub-assemblies do not have a Configurations category node; however, the activate configuration name is appended to the part name. You can see this illustrated below.

Notice the root assembly is called "FourConnectors_w_Leads - Female Mixed Short." (FourConnectors_w_Leads is the name of the assembly model file, and Female Mixed Short is the name of the active configuration.)

There is a subassembly called "Four Connectors<1> - Male Large." This means that Male Large is the active configuration of the model Four Connectors in this assembly. You can see the Configurations category under this component, which lets you know you can change the active configuration from Male Large to something else if desired.

The Four Connectors subassembly has several parts under it. You will see one called "Male Connector<1> - Large." This tells you that Large is the active configuration of the part called Male Connector. Under that component, there is no configuration category. Since this component is nested in a subassembly of the root, you cannot change the active configuration here.



➢ ***To change the configuration that is used for a component of a subassembly***

If you need to edit which configuration is active in a nested component that is not an immediate child of the root assembly, you must open the subassembly and change the active configuration there.

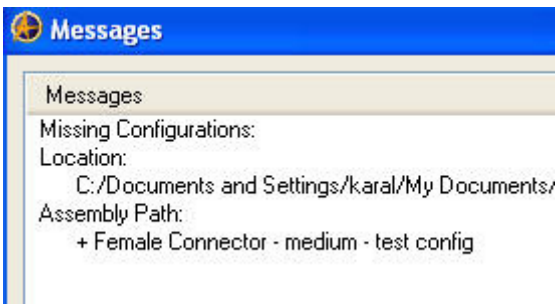
You can open the subassembly separately, or you can edit it in the context of the root assembly.

If a part is edited in the context of an assembly, the part's active configuration will automatically be set to match the configuration held by the assembly.

10.3.2 Missing Design Configurations

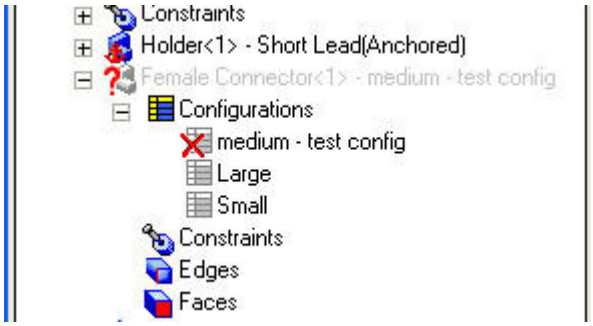
When using configurations, you must use care when deleting configurations that you have created, as they may be used by assembly components.

When you open an assembly in which one of the components is missing its configuration, you will see a message similar to the following:



You can select the **OK** button in the Message dialog, and the assembly will continue to open. You will see the following changes take place in the Design Explorer:

- The component name appears dimmed, with a question mark over the icon.
- The missing configuration name is present in the Configurations tree, with an X over the icon.
- No configuration is marked as active (all configuration icons are in the inactive state).
- No representation of the component with the missing configuration is shown in the work area.



Note: If a configuration is missing from a component in a subassembly of the root assembly, only the first indicator listed above will be seen for that component - the component name will appear dimmed, and will have a question mark over the icon. No Configurations tree will appear under the component.

10.3.3 Using Configurations in Assembly Patterns

Inserting a part or subassembly pattern of a model with multiple configurations

When you insert a part/subassembly pattern in an assembly workspace, the configuration of all instances of the pattern will match the configuration of the seed (the original instance) at the time of creation.

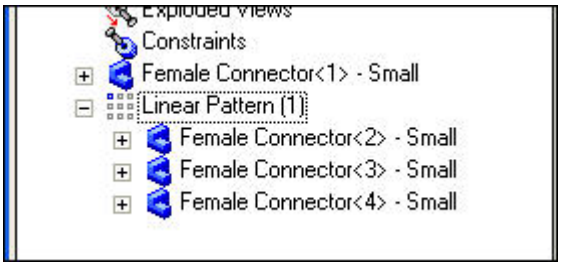


Figure 79: Design Explorer Showing Pattern as Created

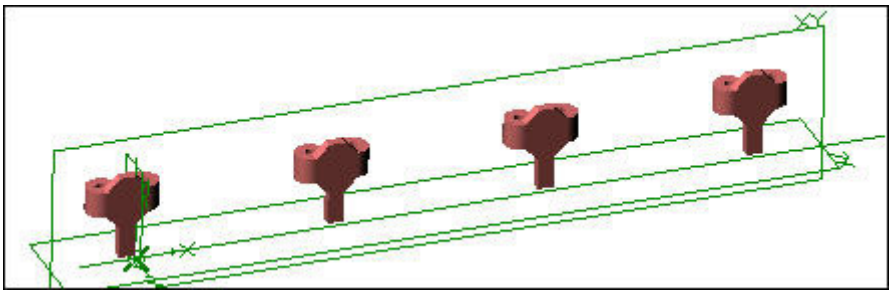


Figure 80: Pattern displayed in the work area - all instances are the same configuration

Once the pattern has been created, you can change any of the instances individually to a different configuration.

➢ ***To change an instance in an assembly pattern:***

1. Click the plus symbol next to the instance that you want to change. This will expand the tree to show the configurations available.

2. Double-click the configuration you want displayed for that instance to make it the active one.

   The model will update in the work area to reflect the new configuration.

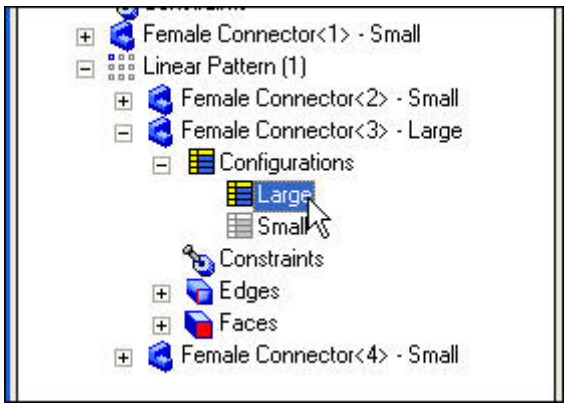


Figure 81: Design Explorer showing pattern when one instance is changed

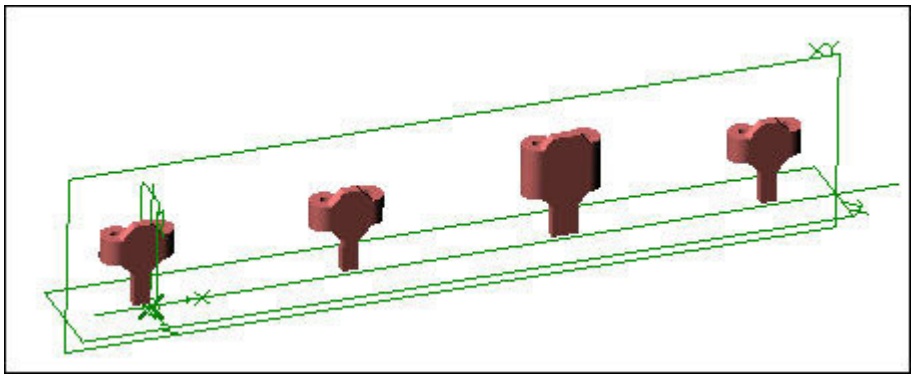


Figure 82: Pattern displayed in the work area - one instance displays a different configuration

Behavior of pattern instances in different configurations

The number of instances in a pattern is a parameter. How the instances behave in different configurations depends on the lock properties of each configuration.

Example 1:

The example assembly above has 1 configuration - Config 1. Assembly Config 1 has a pattern of 4 Female Connectors.

Create another configuration of the assembly - Config 2. Lock all options except Parameter Values, which will be unlocked. Assembly Config 2 is, at this point, is identical to Assembly Config 1.

Next, activate Assembly Config 1, and edit the pattern so that it has seven instances instead of four. Then, activate Assembly Config 2. In the Design Explorer, the Female Connector pattern will show the new instances, but they will be suppressed. This is because the option Part/Subassembly Suppression is locked, which means all new entities be suppressed in Config 2. However, **Parameters** is unlocked, which means the number of instances follows the changes made in the active configuration (Config 1 in this example).

The result will be:

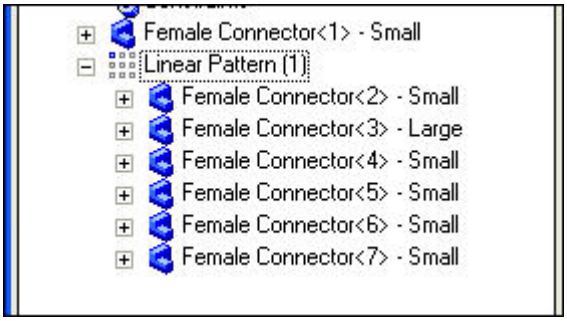


Figure 83: Design Explorer of Assembly Config 1

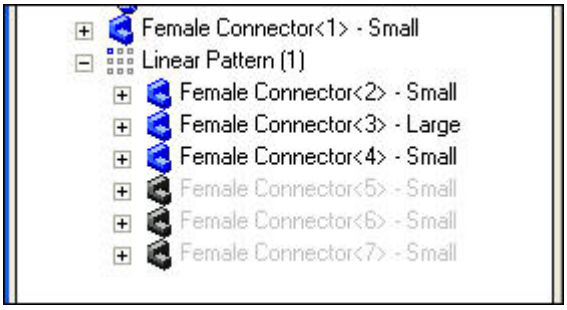


Figure 84: Design Explorer of Assembly Config 2 - new pattern instances suppressed

Example 2:

Using the first example assembly again: Assembly Config 1 has a pattern of four Female Connectors.

Create another configuration of the assembly - Config 2. This time, lock all options. Assembly Config 2 is, at this point, identical to Assembly Config 1.

Next, activate Assembly Config 1, and edit the pattern so that it has seven instances instead of four. Then, activate Assembly Config 2. In the Design Explorer, the Female Connector pattern will NOT show the new instances at all. This is because both Part/Subassembly Suppression and Parameter Values are locked.

The result will be:

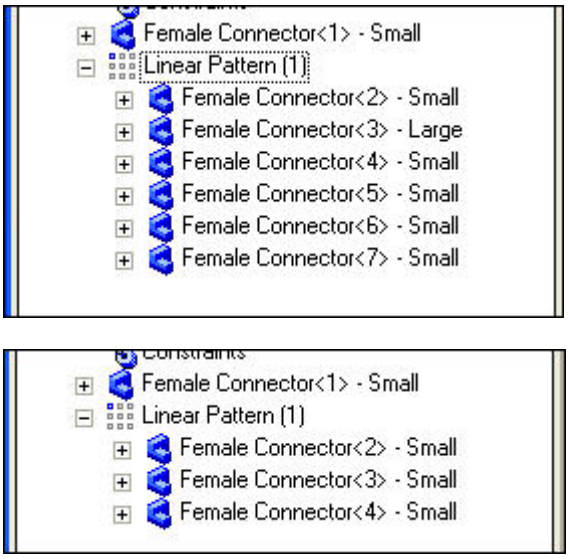


Figure 85: Design Explorer of Assembly Config 2 - new pattern instances not included

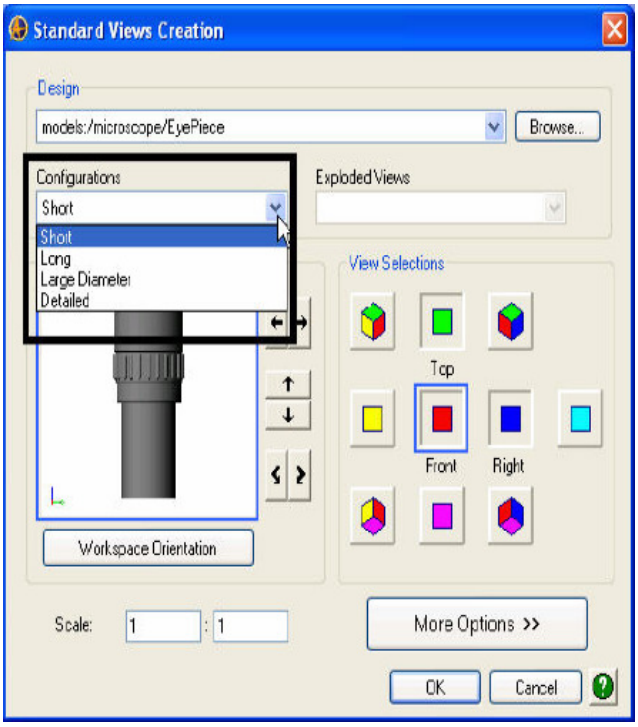
10.3.4 Helpful Notes on Design Configurations in Assemblies

- All configurations have the same set of constraints. A constraint added to one configuration will be added to all configurations. However, constraint values are parameters that can differ between configurations.
- All configurations have a common set of available components. A component deleted from one configuration is deleted from all configurations; however, you can suppress components in individual configurations.
- In the Design Explorer, configurations are listed in the order they were created.
- You cannot delete the active configuration. If only a single configuration exists, it is by default the active configuration, and the **Delete** option will be disabled.
- If you change configurations in the assembly while editing an exploded view, that configuration will open in the exploded view the next time you go back to it.

10.4 Using Configurations in Drawings

When inserting new standard drawing views, you can choose which configuration you would like to see in the view.

The Standard Views Creation dialog has a field called Configurations with a drop-down menu for you to select the configuration you want to see in the views. Since all models contain at least one configuration (Config<1>) by default, the Configurations field will always be populated with a selection. If your model contains more than one configuration, you can select the one you want from the menu.



Notes:

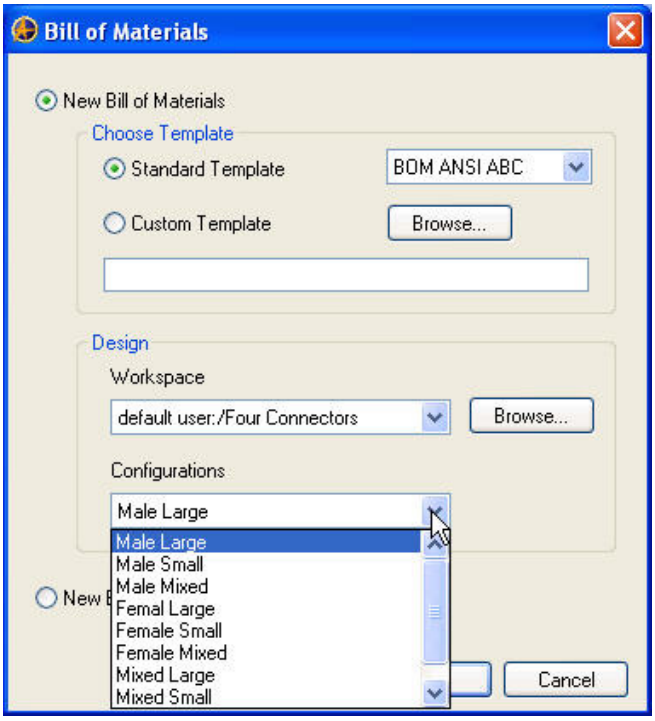
You cannot create drawing views of assemblies with missing configurations.

If you have an assembly that includes BOTH design configurations AND *inter-design constraints* (on page 218) in the same assembly or subassembly, you will not be able to use Fast View mode when creating drawing views for the assembly.

10.5 Using Configurations in a BOM

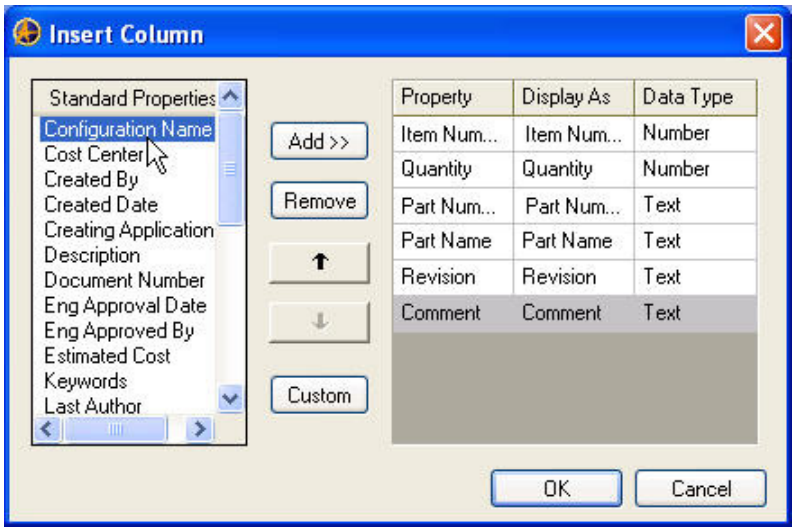
When creating a BOM for a design, you can choose which configuration you would like to include.

The Bill of Materials dialog has a field called Configurations with a drop-down menu for you to select the configuration.



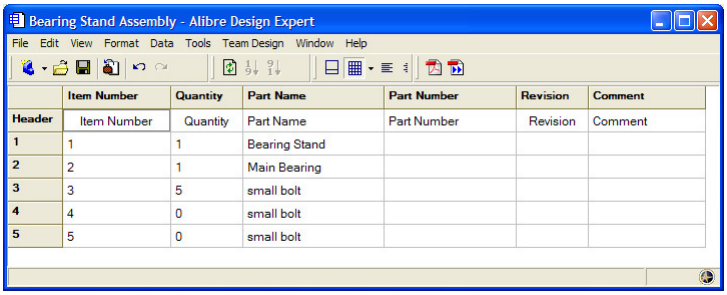
Two standard properties are added to the available options for inserting a column in a BOM. They are:

- Configuration Name - The name of the active configuration of a part used in an assembly.
- Part-Configuration Name - The name of the active configuration appended to the name of the design.



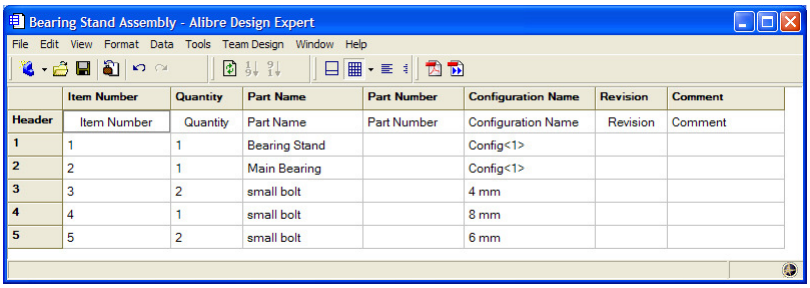
Below are two examples of BOMs that were created for an assembly that calls out a bolt that has three different configurations. The assembly uses at least one of each bolt configuration.

Example 1: The BOM does not have the column Configuration Name included in it. You can see that the part Small Bolt is listed three times, but the total quantity is listed by the first Small Bolt, and there is no indication given of which configuration is used.



	Item Number	Quantity	Part Name	Part Number	Revision	Comment
Header	Item Number	Quantity	Part Name	Part Number	Revision	Comment
1	1	1	Bearing Stand			
2	2	1	Main Bearing			
3	3	5	small bolt			
4	4	0	small bolt			
5	5	0	small bolt			

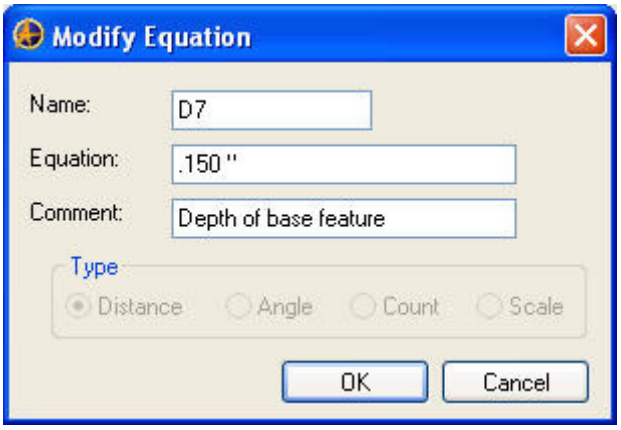
Example 2: The Configuration Name column has been added to the BOM. Now you can see that, again, the Small Bolt part is listed three times, but the total quantity is split up correctly between the configurations used.



	Item Number	Quantity	Part Name	Part Number	Configuration Name	Revision	Comment
Header	Item Number	Quantity	Part Name	Part Number	Configuration Name	Revision	Comment
1	1	1	Bearing Stand		Config<1>		
2	2	1	Main Bearing		Config<1>		
3	3	2	small bolt		4 mm		
4	4	1	small bolt		8 mm		
5	5	2	small bolt		6 mm		

10.6 Using the Equation Editor with Configurations

Although all configurations of a design have a common set of dimensions, the values of those dimensions are parameters whose values can vary from configuration to configuration.



For each dimension, the Name and Comment fields will be the same between configurations. The Equation (which is either an equation or the specific value of the dimension) field can vary between configurations.

You can edit the value of a dimension from the Equation Editor or by double-clicking the value in the sketch and typing in the new value.

CHAPTER 11

Assembly Design

You can create multi-part assemblies of varying function and complexity.

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Importing Parts into an Assembly 231
Joining Parts & Removing Material in an Assembly 231

11.1 Assembly Design Methodology

You can use two distinct assembly design methods, or a combination of both. The first method, often referred to as **bottom-up design**, involves creating each assembly part in an individual part workspace. After the parts have all been individually modeled, you can then insert them into an assembly workspace, and subsequently position and mate them correctly by inserting assembly constraints.

The second method, often referred to as **top-down design**, involves creating all the assembly parts in the assembly workspace. Using this method enables you to design parts while referencing other assembly parts.

Both methods have disadvantages and advantages. The bottom-up design methodology is perhaps the simpler of the two, and enables you to manage the design more efficiently. The top-down design is somewhat more complex, but is valuable when the design of one part is heavily dependent on other parts.

You can also use a combination of the bottom-up and top-down design methods.

11.2 The Assembly Design Interface

Assemblies are designed in assembly workspaces. An assembly can be comprised of parts and other assemblies, referred to as **subassemblies**. The parts and subassemblies that constitute an assembly are referred to as **constituents**.

A typical assembly could have the following structure:

Top-Level Assembly

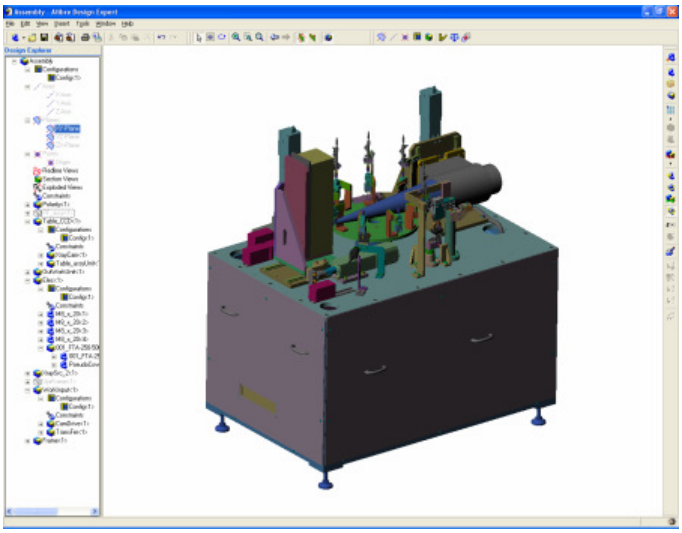
Subassembly A

- Part A1
- Part A2
- Part A3

Subassembly B

- Part B1

Part B2 Part C Part D



Assembly Workspace

The assembly workspace looks similar to the part workspace. However, the Sketching and Part Modeling toolbars are not displayed in the assembly workspace. Instead, the **Assembly Modeling** toolbar is displayed by default on the right side of the workspace.

Insert Part/Subassembly ... insert a part into the assembly

Insert New Part ... insert and begin designing a new part

Insert New Sheet Metal Part ... Insert and begin designing a new sheet metal part

Insert New Assembly ... Insert and begin designing a new assembly

Insert Pattern ... pattern a part in the assembly

Insert Duplicate ... insert a copy of a part in the assembly

Edit Part/Subassembly ... edit a part or subassembly

Quick Constraint Mode ... drag and drop constraint placement

Insert Assembly Constraint ... manually insert an assembly constraint

Move Part ... move an individual part or parts

Rotate Part ... rotate an individual part or parts

Precise Placement ... position a part precisely

Minimum Motion ... use with Rotate or Move to localize motion to selected part

Equation Editor

Color Properties ... modify color properties of parts

Regenerate ... update assembly constraints

Manual Explode ... explode the parts manually

Auto Explode ... allow the software to explode the parts automatically

Expand Explosion ... increase the distance parts explode (auto mode)

Contract Explosion ... decease the distance parts explode (auto mode)

View Part Trails ... show/hide the trail lines

Edit Exploded View Steps ... view/edit exploded view steps

In assembly workspaces, the top-level assembly is listed first in the Design Explorer. Parts and subassemblies are listed at the bottom in the order in which they are inserted or created. The assembly icon signifies an assembly or subassembly item. The part icon signifies a part item.

The Design Explorer also lists the assembly reference geometry, assembly constraints, redline views, section views, as well as the faces and edges associated with each part.

You can click the plus ⊞ sign next to an item to expand it and see its associated details. For example, you will see a subassembly's constituents if you expand it.

You can insert the same part or subassembly multiple times into an assembly. When inserting duplicate parts or subassemblies, the item will be listed in the Design Explorer with its original name followed by a numeric label to indicate how many instances have been inserted, e.g. **Part1<1>**, **Part1<2>**, **Part1<3>**, etc.

11.3 Assembly Basics

You can create an assembly by inserting parts you have already designed, inserting imported parts, or designing new parts in the context of the assembly.

11.3.1 Opening a New Assembly and Inserting Existing Parts

➢ ***To open a new assembly and insert existing parts:***

1. Open a new assembly workspace. The assembly workspace and the **Insert Part/Subassembly** dialog appear.
2. In the **Insert Part/Subassembly** dialog, select the part to be inserted into the assembly.
3. Select the components to insert into the assembly. Press the **Ctrl** key as you select multiple components. To select a series of items, hold the **Shift** key and select the first and last parts in the series.

Note: To start with a blank assembly workspace, click **Cancel** on the **Insert Part/Subassembly** dialog.

4. Click **OK** to insert the part. A preview of the part(s) appears in the assembly workspace. The **Inserting** dialog also appears.
5. Move the cursor to move the part(s), if necessary.
6. Click once to place the part(s) in the workspace.
7. If necessary, continue to click to insert duplicates.
8. Press **Esc** or click **Finish** in the **Inserting** dialog to complete the insertion.

11.3.2 Anchored Parts

A part's position in an assembly can be fixed by **anchoring** the part in the work area. An anchored part cannot be moved. When a part is anchored, the **Anchor** icon is displayed on the part in the Design Explorer. You can anchor any part, as well as remove the anchor state from a part, at any time.

➢ ***To add or remove the anchor state:***

Right-click a part in the Design Explorer and select **Anchor Part** from the pop-up menu.

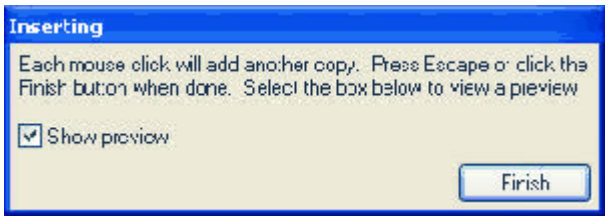
11.3.3 Inserting an Existing Design Into an Open Assembly

You can insert parts or sub-assemblies any time after an assembly has been opened.

➢ ***To insert an existing design into an assembly:***

1. From the **Insert** menu, select **Part/Subassembly**; or right-click in the work area and select **Insert Part/Subassembly** from the pop-up menu; or press **Ctrl + Shift + I** on the keyboard.

   The **Insert Part/Subassembly** dialog appears.
2. In the **Insert Part/Subassembly** dialog, select a design to be inserted into the assembly. You can also select multiple items to be inserted simultaneously. Hold the **Ctrl** key when selecting multiple items.
3. Click **OK** to insert the design. A preview of the design(s) appears in the assembly workspace. The **Inserting** dialog appears.


Inserting
Each mouse click will add another copy. Press Escape or click the Finish button when done. Select the box below to view a preview
Show preview
Finish

4. To insert the design so that the design's origin is initially coincident with the assembly's origin, deselect the **Show preview** option.
5. If the **Show preview** option is left on, you can move the cursor to position the design(s) at any location if necessary.

6. Click once to place the design(s) in the workspace.

7. If necessary, continue to click to insert duplicates.

8. Press **Esc** or click **Finish** in the **Inserting** dialog to complete the insertion.

11.3.4 Selecting Parts in the Assembly

You can select individual parts in the assembly from the Design Explorer or the work area. When you move your cursor over a part in the Design Explorer, the part is highlighted in the work area. When you select a part in the Design Explorer, the part is selected and highlighted in the work area, and vice versa.

➢ ***To select a part in the work area:***

1. Hold the **Ctrl** key and move the cursor over the part you want to select. The part highlights and the cursor changes to .

2. Click to select the part. The part is highlighted in the work area as well as the Design Explorer.

11.3.5 Part Display Options

You can select from four different display modes to control how parts are displayed.

The default display type is **Shaded**. Other display options include **Wireframe**, **Shaded & Visible Edges**, and **Shaded & All Edges**.

➢ ***To change the display of all items in the assembly:***

From the View menu, select **Display** and one of the four options:

- **Shaded:** displays parts in shaded mode, edges are not outlined.
- **Wireframe:** displays parts in wireframe mode, only edges are outlined and displayed.

Note: When viewing the display in wireframe, you can turn the silhouette edges of the model on or off. To do this, from the **View** menu, select **Display**. Check the **Silhouette Edges** option to turn them on. You can also use the tool to toggle them on and off.

- **Shaded & Visible Edges:** displays parts in shaded mode, only visible edges are outlined.
- **Shaded & All Edges:** displays parts in shaded mode, visible as well as hidden edges are outlined.

The display is updated.

Note: You can quickly change between shaded and wireframe display modes by selecting the Shaded and Wireframe tools from the Options drop down list on the Visibility toolbar.

➢ ***To change the display of individual parts:***

1. Right-click on any part.

2. Select **Render Mode** from the popup menu.

3. Select the render mode you want for that part.

11.3.6 Inserting a Duplicate Design Into an Open Assembly

You can insert copies of a part or subassembly that you have already inserted into an assembly.

➢ ***To insert duplicates into an assembly:***

1. Select the design you want to duplicate and insert. You can select the design in the Design Explorer or work area.
2. From the **Insert** menu select **Duplicate**. A preview of the part appears in the assembly workspace. The **Duplicating** dialog also appears.
3. Move the cursor to move the design, if necessary.
4. Click once to place the design in the workspace.
5. If necessary, continue to click to insert additional duplicates.
6. Press **Esc** or click **Finish** in the **Duplicating** dialog to complete the insertion.
7. The duplicate appears and is listed in the Design Explorer.

OR

1. Left click on a part and hold down the left mouse button.
2. Hold down the CTRL key.
3. Drag the part. A duplicate is instantly inserted, and you can drag it around the assembly.
4. To finalize the duplicate, release the left mouse button.

11.3.7 Inserting a Pattern of Parts in an Assembly

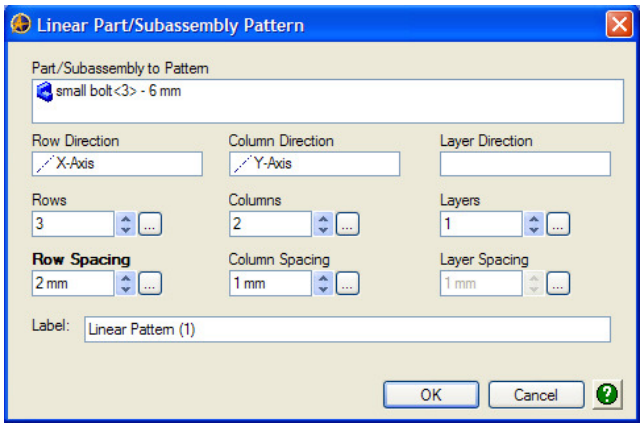
You can pattern a part or subassembly that you have already inserted into an assembly.

Linear Pattern

You can use a linear pattern to repeat a part in one, two, or three linear directions.

➢ ***To create a linear pattern of a part in an assembly:***

1. From the **Insert** menu, select **Part/Subassembly Pattern > Linear**. The **Linear Part/Subassembly Pattern** dialog box appears.



2. In **Part/Subassembly to Pattern**, select the part you want to

3. In **Row Direction**, select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the row.

4. In **Column Direction**, select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the column.

5. In **Layer Direction,** select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the layer.

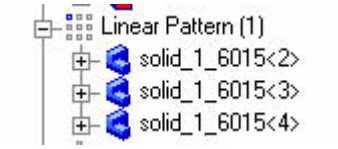
6. Enter the number of copies you want in each direction, including the original. A value of 1 will make no additional copies in that direction.

7. Enter the distance you want between each copy in the spacing fields.

8. In **Label**, enter a unique name for the pattern.

9. Choose **OK**.

The linear pattern will appear in the Design Explorer, with the parts in the pattern listed under it in the tree. These patterned parts cannot be edited or moved, however, any of the patterned parts can be deleted by right-clicking them in the Design Explorer and choosing **Delete**.

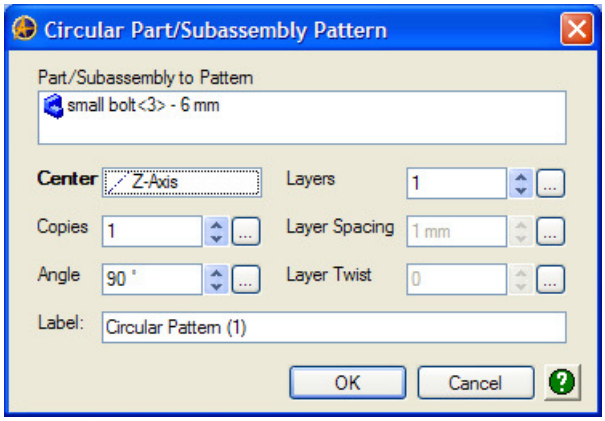


Circular Pattern

You can use a circular pattern to repeat a part in a radial direction around a centerline.

➢ ***To create a circular pattern of a part in an assembly:***

1. From the **Insert** menu, select **Part/Subassembly Pattern > Circular**. The **Circular Part/Subassembly Pattern** dialog appears.



2. In **Part/Subassembly to Pattern**, select the part you want to pattern.

3. In **Center**, select the axis you want the parts to be patterned around from the Design Explorer.

4. In **Copies**, enter the number of copies you want, including the original.

5. In **Angle**, set the angle value.

6. In **Layers**, enter the number of layers you want in the direction of your centerline. The same number of copies will be created on each layer.

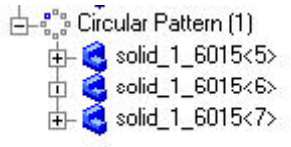
7. In **Layer Spacing**, set the distance between each layer

8. In **Layer Twist**, set an angle to rotate the layers with respect to each other, if desired.

9. In **Label**, enter a unique name for the pattern

10. Click **OK**.

The circular pattern will appear in the Design Explorer, with the parts in the pattern listed under it in the tree. These patterned parts cannot be edited or moved, however, any of the patterned parts can be deleted by right-clicking them in the Design Explorer and choosing **Delete**.



11.3.8 Moving and Rotating Parts Freely

You can move or rotate a part freely as long as it is not anchored or constrained in such a way that limits movement.

➢ ***To move a part:***

1. Left-click a part and drag it while continuing to hold down the left mouse button. The part moves as you drag the cursor. You can release the **Ctrl** key after the part begins to move.

   Or

   Select the **Move Part** tool from the Assembly Modeling toolbar.

   Or

   From the **Tools** menu select **Move Part**.

2. Move the cursor over the part you want to move.

3. Click and drag the cursor to move the part.

➢ ***To rotate a part:***

1. Move the cursor over the part to rotate.

2. Hold the **Ctrl** key, select a part in the work area holding both mouse buttons, and drag the cursor. The part rotates as you drag the cursor. You can release the **Ctrl** key after the part begins to rotate.

   Or

   Select the **Rotate Part** tool from the Assembly Modeling toolbar.

   Or

   From the **Tools** menu select **Rotate Part**.

3. Move the cursor over the part you want to rotate.

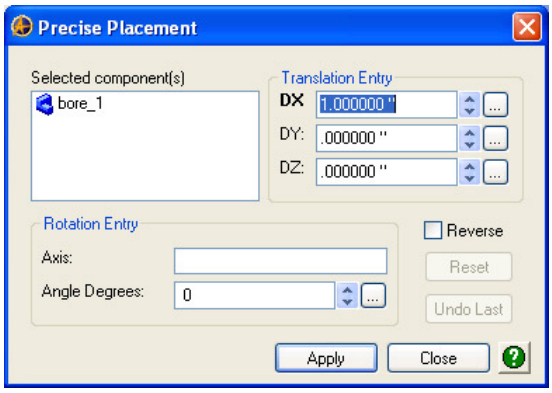
4. Click and drag the cursor to rotate the part.

11.3.9 Moving and Rotating Parts Precisely

You can move or rotate a part precisely, if necessary, as long as a part is not anchored or constrained in such a way that limits movement.

➢ ***To move or rotate a part a precise distance or angle:***

1. Select the **Precise Placement** tool from the Assembly Modeling toolbar; or from the **Tools** menu select **Precise Placement**. The **Precise Placement** dialog appears.



2. Select the part to move or rotate.

3. To move a part, in the **Translation Entry** area, specify the relative translation distance by entering values for **DX**, **DY**, and **DZ**.

   Or

   To rotate a part, in the **Rotation Entry** area, select the rotation **Axis** and specify the relative rotation **Angle** in degrees. The rotation axis can be an axis or linear edge on a part.

4. Click **Apply** to move or rotate the part.

5. Click **Close** to close the dialog.

11.3.10 Moving Parts to Simulate Assembly Physical Motion

➢ ***To position a part or subassembly***

1. While holding the left mouse button down, drag the part or subassembly to the new position.

   Or

   From the **Tools** menu, select **Move Part**.

2. Click the **Minimum Motion** tool on the Assembly Modeling toolbar

3. Click and drag the part or subassembly. The part or subassembly moves a minimum number of parts.

11.3.11 Hiding a Part

You can hide individual parts in an assembly. A part that is hidden still fully participates in the regeneration of the model, as well as all physical property calculations. It is simply not visible.

➢ ***To hide an individual part:***

Right-click a part in the Design Explorer and select **Hide** from the pop-up menu; or right-click a part in the work area and select **Hide** from the pop-up menu. The part is hidden in the work area and is listed in gray in the Design Explorer.

➢ ***To unhide the part:***

Right-click the part in the Design Explorer and select **Hide** again.

11.3.12 Suppressing a Part

You can suppress individual parts in an assembly. A part that is suppressed does not participate in the regeneration of the model, or any physical property calculations.

➢ ***To suppress an individual part:***

Right-click a part in the Design Explorer and select **Suppress** from the pop-up menu; or right-click a part in the work area and select **Suppress** from the pop-up menu. The part is hidden in the work area and is listed in gray in the Design Explorer.

➢ ***To unsuppress the part:***

Right-click the part in the Design Explorer and select **Suppress** again.

11.3.13 Applying Color Properties to a Part

You can apply different color properties to each part in an assembly.

➢ ***To apply color properties to a part:***

1. Right-click a part in the work area or Design Explorer.
2. Select **Color Properties**.
3. Select a color and set the **Reflectivity** and **Opacity** levels as necessary.
4. Click **OK**.

OR

1. Select a part either in the Design Explorer or the work area.
2. Select the **Color Properties** tool from the Part Modeling toolbar; or right-click in the work area and select **Color Properties** from the pop-up menu; or from the **Edit** menu, select **Color <Part Name> Properties**. The **Color Properties** dialog appears.
3. Select a color and set the **Reflectivity** and **Opacity** levels as necessary.

4. Click **OK** to apply the properties.

Note: You can apply the same color properties to the entire assembly. Select the top-level assembly in the Design Explorer, or from the **Edit** menu select **Select All**. Follow steps 1-4 above to apply the color properties.

11.3.14 Checking Measurements

You can check measurements on an individual part basis or the assembly basis.

- ***To check part/assembly measurements:***

1. Right-click a part or assembly in the Design Explorer and select **3D Measurement Tool** from the pop-up menu

   Or

   Select a part in the Design Explorer or work area, right-click and select **Properties** from the pop-up menu.

   The **Measurement Tool** dialog appears and lists the physical properties for the selection.

2. Click **Close** on the dialog when finished viewing the properties.

11.3.15 Viewing Part Reference Geometry

You can view the reference geometry of a part while in an assembly. These features can then be used to constrain the part to the assembly, if desired.

- ***To view part reference geometry:***

1. Right-click the part in the Design Explorer or the work area.

2. Choose **Show Reference Geometry**. The part reference features will appear in the design explorer and in the model window.

- ***To hide part reference geometry***

1. Right-click the part in the Design Explorer or the work area.

2. Choose **Show Reference Geometry.** (This will be checked if reference geometry is shown. Choosing it again will uncheck the option, hiding the reference geometry).

11.4 Assembly Constraints

You can insert assembly constraints to precisely position and mate parts with respect to each other in an assembly. Assembly constraints also dictate how parts move or rotate with respect to other parts.

A part is initially unconstrained and has six degrees of freedom when first inserted into an assembly. You can move or rotate an unconstrained part in any direction. As you place assembly constraints on a part, the degrees of freedom are reduced and you begin to limit how a part can be moved or rotated. A fully constrained part has zero degrees of freedom and its movement and/or rotation depend on the movement/rotation of the part or parts to which it is constrained.

11.4.1 Assembly Constraint Types

You can use five different types of assembly constraints: **mate**, **orient**, **angle**, **align**, and **tangent**. Each constraint type is valid for specific combinations of items. You can apply constraints to the following items:

- Reference planes and axes
- Linear edges
- Planar faces
- Cylindrical faces
- Spherical, conical, and toroidal faces

You will always select two items when applying a constraint. The following table summarizes which constraint types can be applied to various combinations of items.

	Plane	Cylinder	Line	Sphere
Plane (planar face or reference plane)	Mate Orient Angle Align			
Cylinder (cylindrical face)	Tangent Align	Align Tangent Orient		
Line (linear edge or axis)	Align Orient	Align Tangent	Align Orient	
Sphere (spherical face)	Tangent	Align Tangent	Align	Align Tangent

11.4.2 Inserting Assembly Constraints - Quick Constraint Mode

Alibre Design has a visual constraint application mode called Quick Constraint Mode. Quick Constraint Mode allows you to focus on the job of quickly constraining items without knowing explicitly what constraints need to be applied to get the orientation and behavior you want. This is accomplished through visual feedback and intelligence built into the constraint application system.

Two workflows can be used when in Quick Constraint mode:

- Drag and Drop
- Explicit selection

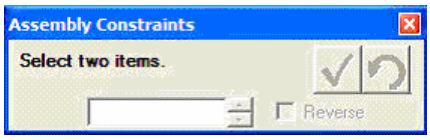
Note: Quick Constraint mode is only used to apply constraints, not to edit previously created constraints.

Drag and Drop Constraint Placement

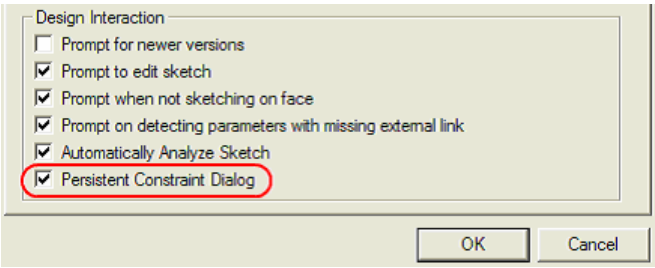
➢ ***To enter Quick Constraint mode***

1. Press **Ctrl + Q** or select the **Quick Constraint** tool from the modeling toolbar.

   The Quick Constraint button is depressed on the modeling toolbar and the dialog appears.

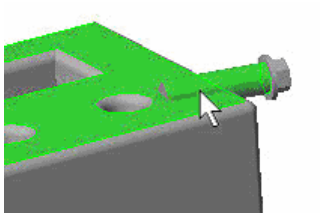


Note: You can choose whether to display the dialog at all times or only after two items to constrain have been selected. If the dialog does not display immediately and you would like it to, select the option **Persistent Constraint Dialog** from the **Tools > Options > General tab**.

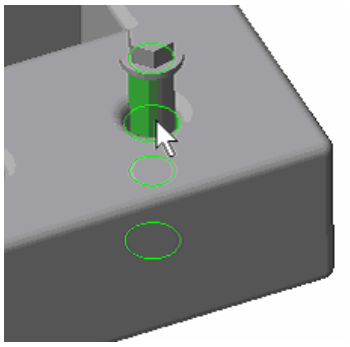


2. Click on the item you want to constrain and begin to drag it. Clicking and dragging on a face or edge will make it become green. This signifies that you are in the process of a drag and drop constraint.

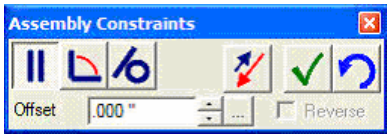
3. While dragging the part, hover the mouse over another part. When the mouse hovers over an area of another part, the area it is over will also turn green. When this happens, a constraint will automatically be previewed. In this case, an Align constraint is applied which makes the axis of the cylinder and the face of the other part parallel.



4. Continue dragging the first part around the second part. Whatever part you hover over will change and, as it does, so will the constraint preview. In this example, we want to insert the bolt into the hole. While dragging the bolt, we will hover the mouse over the interior cylindrical face. The bolt will orient itself appropriately when we do this, and the interior cylindrical face turns green (hidden).



5. When the constraint preview appears as you want, release the mouse button. The dialog will now populate with options that are relevant to the two types of geometry that you are constraining. In this case, we can make the items parallel, set an angle, create an offset, or flip the bolt in the other direction.



6. When you are ready to apply the constraint, click the check mark. To cancel, click the **Cancel** button. You can now begin the process again.

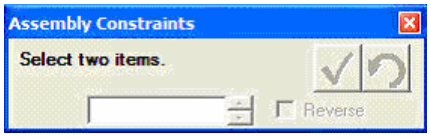
Note: You can only use drag and drop constraints if the item you are dragging is completely unconstrained. After you apply the first constraint to it, you will need to continue with the second mode shown below.

Explicit Selection Constraint Placement

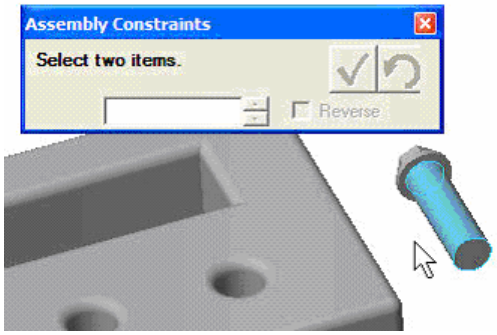
The only difference between this mode and the previous mode is that you do not drag items. Instead, you explicitly click the two items you want to constrain. We will show this using the previous example.

➢ ***To enter Quick Constraint mode***

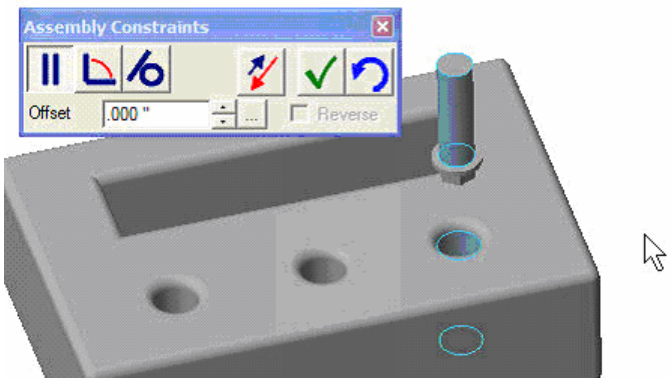
1. Press **Ctrl + Q** or select the **Quick Constraint** tool from the modeling toolbar. The Quick Constraint button is depressed on the modeling toolbar and the dialog appears.



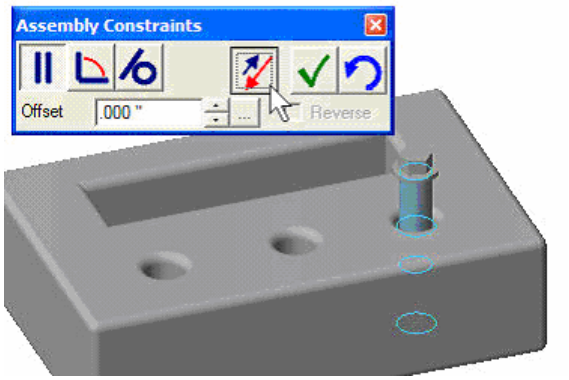
2. Select the first item with a single click. This can be a face, edge, axis, plane, or point. The item you click will highlight. In the example below, we have clicked on the cylindrical face of the bolt.



3. Click the second item. This can also be a face, edge, axis, plane, or point. The second item highlights as well, and the constraint is previewed. The dialog will now populate with options that are relevant to the two types of geometry that you are constraining. In this case, we can set an angle, create an offset, or flip the bolt in the other direction. In this case, we can see the bolt is facing in the wrong direction.



4. Select the **Flip** button, if necessary. This flips the orientation of the parts. Now, our bolt is in the correct orientation.



5. By choosing the different buttons on the toolbar, you can switch the type of constraint. In this case, we can choose to make the two cylinders tangent, parallel, or at an angle to each other.

6. When satisfied with the constraint, click the check mark. Or, select **Cancel** to exit out without committing the constraint.

Note: After you are satisfied with the preview, you can immediately begin clicking on new items to constrain. When you click on a new item while a previous constraint is being previewed, the previewed constraint becomes committed, and you can continue making new constraints. Using this method, you can very quickly apply constraints without needing to explicitly apply each one.

11.4.3 Inserting Assembly Constraints - Manual Method

➢ ***To insert assembly constraints:***

1. Click the **Insert Assembly Constraint** tool from the Assembly Modeling toolbar; or right-click in the work area and select **Insert Assembly Constraint** from the pop-up menu; or from the **Insert** menu select **Assembly Constraint**. The **Assembly Constraints** dialog appears.

2. Select the first edge, face, plane, or axis. The selected item is highlighted in the work area and is listed in the **Surfaces to constrain** area.

3. Select the second edge, face, plane, or axis. The selected item is highlighted in the work area and is listed below the first selection in the **Surfaces to constrain** area.

Note: If you prefer, you can pre-select the two entities, and then open the Assembly Constraint dialog. The Surfaces to constrain field will be populated with the selections.

4. Select the appropriate constraint type. Only the constraint types that are valid for the selected items are available.

5. If you are applying the **Angle** constraint, specify the angle in degrees.

6. If you are applying the **Tangent** constraint, select **Inside** or **Outside**.

7. You can specify an **Offset** value if you are applying a **Mate**, **Align**, or **Tangent** constraint.

8. Select the **Preview** option to check the result before applying it. If the preview shows the parts in the incorrect position, modify the constraint properties accordingly.

9. Click **Apply** to finalize the constraint. The second component in the list is constrained to the first component, and the constraint is listed in the Design Explorer under the Constraints node. The constraint label lists the constraint type. Expand the node to view its constrained entities. The dialog remains open so you can create another constraint.

10. Repeat the procedure to create another constraint, or click **Close** to exit the Assembly Constraints dialog.

11.4.4 Inter-Design Constraints

While in part edit mode, if you use a face on an existing part as a sketch plane or to create a new reference plane, an i**nter-design constraint** will automatically be created. This constraint will be displayed in the Design Explorer under the Constraints node when you switch back to assembly edit mode. The inter-design constraint label, by default, lists the two related parts (the parent part and the associated part).

After an inter-design constraint has been created, any change made to the parent part will automatically update the associated part. To break the interdependency, delete the inter-design constraint in assembly edit mode. If you do not want the new part to change as a result of other parts changing, it is good practice to immediately delete the inter-design constraints from the main assembly as soon as you are finished working on the newly created part.

11.4.5 Managing Assembly Constraints

Each new assembly constraint that you insert will be listed under the **Constraints** node in the Design Explorer, and/or under the constituent that it references. You can choose how you would like to display the constraints.

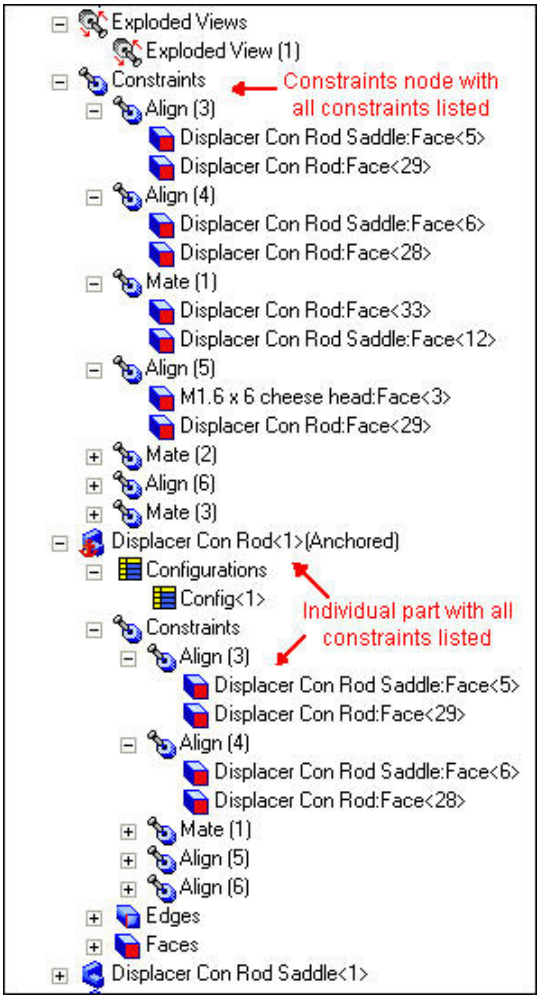


Figure 86: Design Explorer Showing Constraints Listed Under Constraints Node and Individual Components

If you move the cursor over a constraint or select a constraint in the Design Explorer, the applicable entities will highlight in the work area.

➢ ***To change how constraints are viewed:***

1. From the **View** menu, select **Assembly Constraints**.
2. Check **As List** to see the constraints listed in the Design Explorer under the Constraints node. This is a toggle on/off option.
3. Check **With Component** to see the constraints listed under the parts to which they relate. This is a toggle on/off option.

Both options can be checked at the same time, allowing the constraints to be shown in both locations in the design explorer.

➢ ***To change a constraint's properties:***

You can change constraint properties such as offset distance or angle after a constraint has been applied.

1. Right-click the constraint in the Design Explorer and select **Edit** from the pop-up menu. The **Assembly Constraints** dialog appears.
2. Modify the appropriate parameters.
3. Click **OK** to apply the changes.

➢ ***To rename a constraint:***

You can customize the constraint label in the Design Explorer.

1. Right-click the constraint in the Design Explorer and select **Rename** from the pop-up menu; or double-click the constraint with a short pause between clicks. The text cursor appears in the label.

2. Type in a new label.

3. Press **Enter** on the keyboard.

➢ ***To delete a constraint:***

Select the constraint and press **Delete** on the keyboard; or right-click the constraint in the Design Explorer and select **Delete** from the pop-up menu.

The constraint is deleted.

11.4.6 Failed Assembly Constraints

A constraint may fail due to the following reasons:

- Constraint was initially applied incorrectly, e.g. applying an align constraint to a planar face and a non-planar face.
- The constraint creates an over-defined condition.
- A condition changes after the constraint was applied, e.g. deleting a part from an assembly or the geometry in which the constraint applies is modified.

A failed constraint is displayed in italics in the Design Explorer. In the example below, the Align constraint has failed.

It is recommended you resolve a failed constraint as soon as the condition occurs.

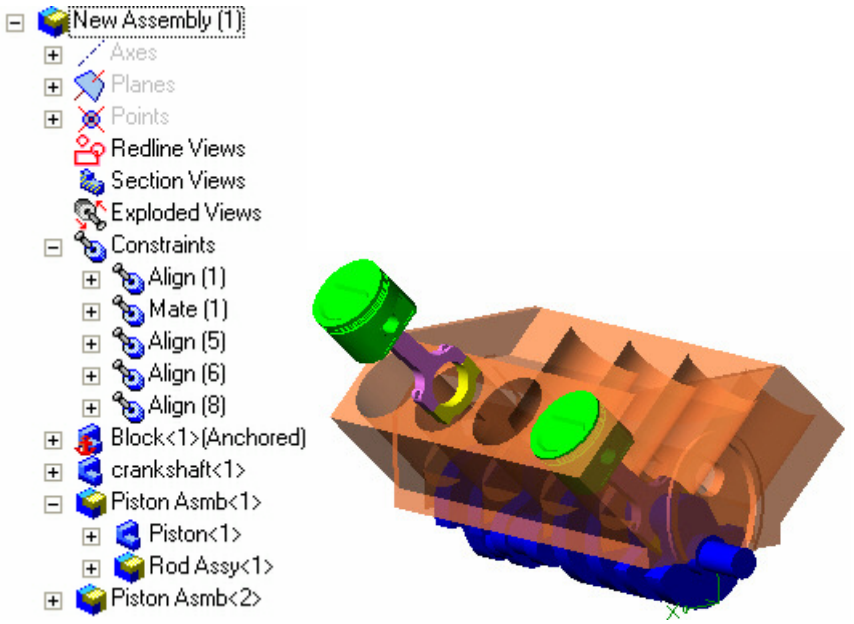
➢ ***To troubleshoot and resolve a failed constraint:***

- Right-click the constraint and select **Status** from the pop-up menu. A dialog appears containing information related to the cause of the constraint failure.
- Right-click the constraint and select **Edit**. The Assembly Constraint dialog appears. Modify the constraint conditions, if applicable.
- In some cases, you may need to delete and reapply the constraint.

11.5 Flexible Subassemblies

By default, subassemblies inserted into an assembly are rigid. This means that a subassembly is treated as a rigid body for purposes of positioning and constraining it in the context of the parent assembly. Consequently, if multiple instances of a subassembly are inserted into an assembly, the components in each instance will be in identical relative positions.

You can easily convert a subassembly from rigid to flexible. When a subassembly is made flexible, all of the parts in that subassembly can be positioned and constrained independently within the context of the top level assembly.

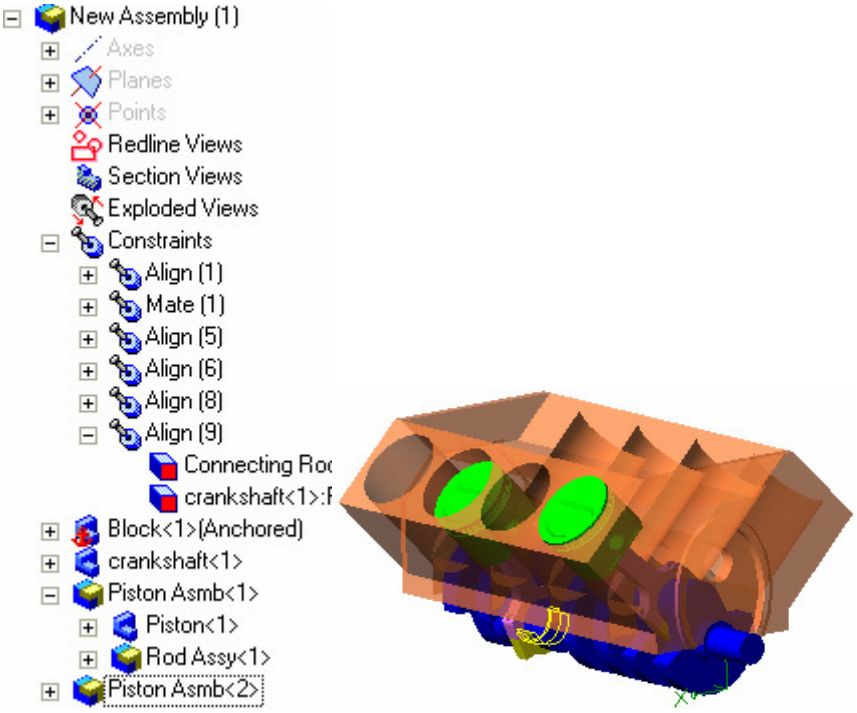


In the example shown here, two instances of a piston assembly are placed in the engine assembly. By default, these subassemblies are rigid. Since each piston subassembly is treated as a rigid body, it is not possible to properly constrain them such that each piston is aligned with its respective cylinder, and each connecting rod is aligned with the crankshaft. This is because in order to completely satisfy the set of constraints, the piston in the second subassembly must move relative to its connecting rod. You can easily accomplish this by making the second subassembly flexible.

➢ ***To make a subassembly flexible:***

Right-click the desired subassembly in the Design Explorer and click the **Make Flexible** toggle on the pop-up menu.

Each part in the subassembly can now be constrained independent of the other parts in the subassembly. In the example above, it is now possible to properly locate each piston subassembly, as shown here:



You can again make a subassembly rigid by turning off its **Make Flexible** toggle.

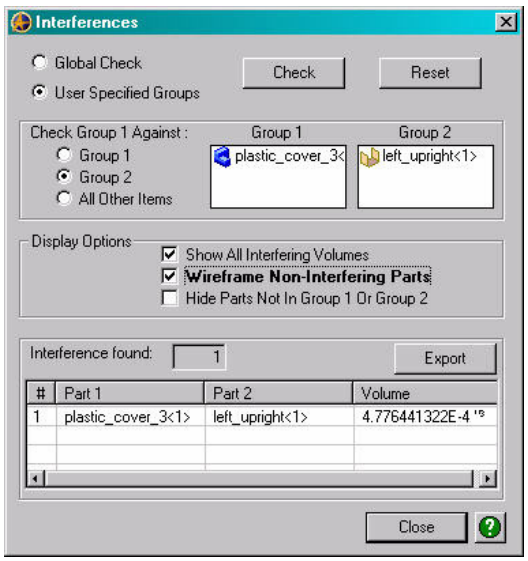
Note: You cannot use the Make Flexible command on any subassembly that contains a pattern.

11.6 Checking for Interferences

You can check for interferences between selected assembly parts, selected groups of parts, or all assembly parts.

➢ ***To check for interferences:***

1. Select the **Check for Interferences** tool from the Inspection toolbar; or from the **Tools** menu select **Check for Interferences**. The **Interferences** dialog appears.



2. Select **Global** or **User Specified Groups**. The **Global** option checks for interferences considering all parts of the assembly. The **User Specified Groups** option checks only the parts you select.

3. If you selected **User Specified Groups**, you must select at least one part in **Group 1.** You can then choose to check for interferences between the selected Group 1 part(s) against:

   - **Group 1**: checks for interferences within the selected **Group 1** parts (requires at least two selections)
   - **Group 2**: checks for interferences between **Group 1** parts and parts selected in **Group 2**
   - **All Other Items**: checks for interferences between **Group 1** parts and all other parts in the assembly

4. Click the **Check** button. The number of interferences detected is listed in the **Interference found** box. Each interference will also be listed individually specifying which parts are interfering as well as the interfering volume.

5. To further analyze interferences, select interference **Display Options**:

   - **Show All Interfering Volumes**: displays the detected interferences in red.
   - **Wireframe Non-Interfering Parts:** displays the parts that are not interfering in wireframe display mode.
   - **Hide Parts Not in Group 1 or Group 2**: hides any parts that were not in the Group 1 or Group 2 selections.

6. To export the interference information to a .CSV file, click the **Export** button.

11.7 Inserting an Exploded View

You can insert exploded views of the assembly during the design process. The exploded view allows you to display a configuration of the assembly with the parts separated. You can use the auto explode mechanism, manually separate parts, or a combination of both. You can create multiple exploded views in an assembly workspace. The exploded views are saved with the design and can subsequently be used as views in drawings.

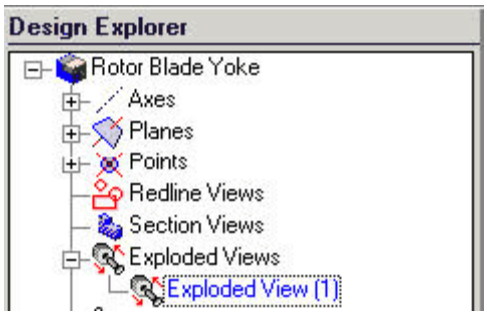
The assembly parts must be constrained before the exploded view can be used. The exploded view however, has no effect on the interpretation of assembly constraints.

11.7.1 Inserting an Exploded View Using Auto Explode Mode

You can create multiple exploded views in an assembly workspace. You can auto explode an assembly to quickly create an exploded view of an assembly. **Mate** and **align** assembly constraints must be applied before an exploded view can be created.

➢ ***To insert an exploded view using Auto Explode mode:***

1. From the **Insert** menu, select **Exploded View**. The assembly changes to the exploded view. An Exploded View item is listed in blue under the **Exploded View** node in the Design Explorer. By default, exploded views will be labeled **Exploded View(1)**, **Exploded View(2)**, etc. You can rename an exploded view, if desired (right-click the view and select **Rename** from the pop-up menu).



The majority of the assembly modeling tools become dimmed and the explode view tools become available on the Assembly Modeling toolbar.

Manual Explode ... explode the parts manually

Auto Explode ... allow the software to explode the parts automatically

Expand Explosion ... increase the distance parts explode (auto mode)

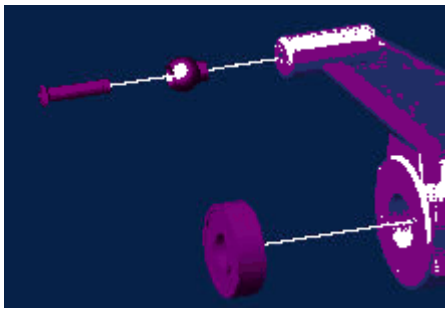
Contract Explosion ... decease the distance parts explode (auto mode)

View Part Trails ... show/hide the trail lines

Edit Exploded View Steps ... view/edit exploded view steps

2. To automatically explode the assembly, click the **Auto Explode Assembly** tool from the Assembly Modeling toolbar; or from the **Tools** menu, select **Auto Explode Assembly**. The assembly is exploded. The distances placed between assembly parts are automatically calculated based on part size and orientation.

By default, trail lines are displayed in the work area which provide a visual guide between a part and the part it is constrained to.



3. To hide the trail lines, deselect the **View Part Trails** tool on the Assembly Modeling toolbar; or from the **View** menu, deselect **Exploded View Trails**.

4. To increase or decrease the explode distance, click the **Expand Explosion** tool (to increase) or the **Contract Explosion** tool (to decrease) from the Assembly Modeling toolbar; or from the **Tools** menu, select **Expand Explosion** or **Contract Explosion**.

5. Continue to expand or contract the exploded view until the desired view is achieved.

6. To restore a part back to its original position, right-click the part and select **Restore To Default Position** from the pop-up menu.

7. To exit the exploded assembly view and return to normal assembly mode, right-click the exploded view in the Design Explorer and select **Exit Exploded View** from the pop-up menu; or from the **Edit** menu, select **Exit Exploded View**; or right-click in the work area and select **Exit Exploded View** from the pop-up menu.

   The assembly is returned to its normal view.

11.7.2 Inserting an Exploded View Using Manual Explode

You can create multiple exploded views in an assembly workspace. You can use the manual explode mode to create a custom exploded view of an assembly. Assembly constraints must be applied before an exploded view can be created. You can use manual explode mode in conjunction with auto explode mode.

➢ ***To insert an exploded view using Manual Explode mode:***

1. From the **Insert** menu, select **Exploded View**. An Exploded View item is listed in blue under the **Exploded View** node in the Design Explorer. By default, exploded views will be labeled **Exploded View(1)**, **Exploded View(2)**, etc. You can rename an exploded view if desired (right-click the view and select **Rename** from the pop-up menu).

2. To enter Manual Explode mode, select the **Manual Explode Mode** tool from the Assembly Modeling toolbar; or right-click in the work area and select **Manual Explode Mode** from the pop-up menu; or from the **Tools** menu, select **Manual Explode Mode**.

3. To specify the explode direction, select an axis, reference plane, model edge, model planar face, or model cylindrical face. You can specify a different explode direction for each part you want to separate. Reference arrows will subsequently be displayed near your selection.

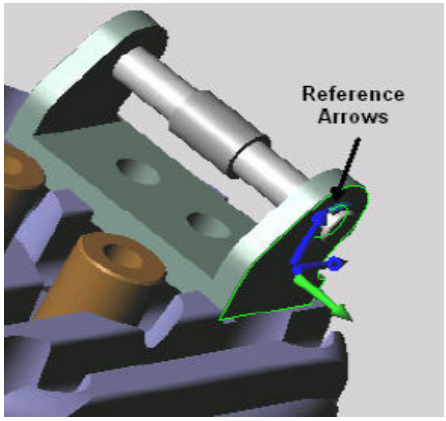


Figure 87: Manual Explode Mode - Displaying Reference Arrows

4. The reference arrows allow you to specify the direction in which you want to manually move a part. The arrow that is currently selected is displayed in yellow and defines the direction the part will move in. To select a different direction, select a different arrow. To remove the reference arrows, click anywhere in any open space in the workspace.

5. Select the part you want to move by left-clicking the part. The part is highlighted. To select multiple parts to move simultaneously, continue to click additional parts while holding the **Shift** key.

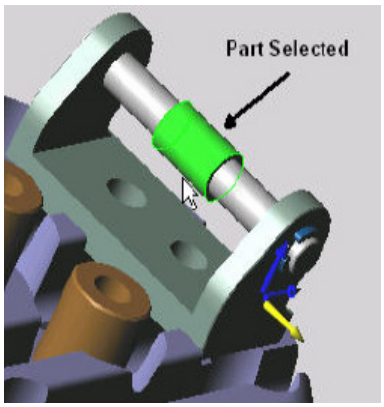


Figure 88: Manual Explode Mode - Displaying Selected Part

6. Left-click again and drag the part(s) in the direction defined by the highlighted reference arrow. You can also move the part(s) in the direction opposite the arrow is pointing.

7. Release the mouse button to complete the separation.

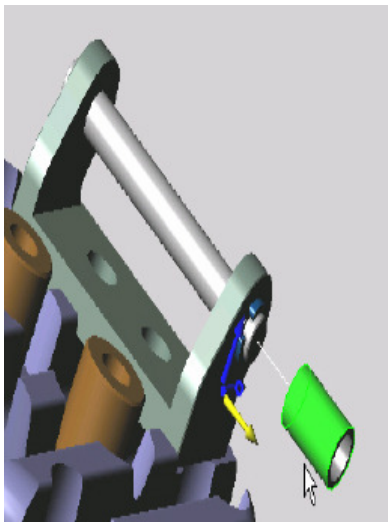


Figure 89: Manual Explode Mode - Displaying Final Part Location

Tip: If you want to move an entire subassembly as one entity, select it in the Design Explorer so that all of the parts in the subassembly highlight. Then left click in the work area and drag the mouse to move the part(s) to the desired location.

8. By default, trail lines are displayed in the work area which provide a visual guide between a part and the part to which it is constrained.

   To hide the trail lines, deselect the **View Part Trails** tool on the Assembly Modeling toolbar; or from the **View** menu, deselect **Exploded View Trails**.

9. To exit Manual Explode Mode, deselect the **Manual Explode Mode** tool from the Assembly Modeling toolbar; or from the **Tools** menu, deselect **Manual Explode Mode**.

10. To automatically increase or decrease the explode distance after moving a part or parts, click the **Expand Explosion** tool (to increase) or the **Contract Explosion** tool (to decrease) from the Assembly Modeling toolbar; or from the Tools menu, select **Expand Explosion** or **Contract Explosion**.

11. Continue to expand or contract the exploded view until the desired view is achieved.

12. To restore a part back to its original position, right-click the part and select **Restore To Default Position** from the pop-up menu.

13. To exit the exploded assembly view and return to normal assembly mode, right-click the exploded view in the Design Explorer and select **Exit Exploded View** from the pop-up menu; or from the **Edit** menu, select **Exit Exploded View**; or right-click in the work area and select **Exit Exploded View** from the pop-up menu.

14. The assembly is returned to its normal view.

11.7.3 Viewing and/or Editing an Exploded View

You can view and edit an exploded view at anytime.

➢ ***To view and/or edit an exploded view:***

1. In the Design Explorer, right-click the Exploded view you want to edit and select **Edit** from the pop-up menu; or select the Exploded view and from the **Edit** menu, select **Edit Exploded View**.

   The Exploded view being edited is listed in blue text in the Design Explorer and is displayed in the work area.

2. Make any necessary changes to the Exploded view.

3. To return to the normal assembly view, right-click the Exploded view being edited and select **Exit Exploded View** from the pop-up menu.

11.7.4 Exploded View Steps

The Exploded View Step Editor allows you to create and manage steps for your exploded view. These steps will be similar to an instruction manual describing the assembling information for the assembly. The steps can be used later when Publishing to PDF to animate the exploded view. (This is available for users with Full Publishing capability only.)

The exploded view steps can be created in two ways:

1. If you choose to auto-explode your assembly, the steps are automatically created.

2. If you choose to manually explode your assembly, a new step is created each time you move a part. However, only one step exists for each part in the assembly, so if you move a part a second time, the step for it updates.

Characteristics of exploded view steps

- Each step will only move a single part from its initial position to its exploded position or from its exploded position to its initial position.
- A step can be moved to a different position in the sequence of steps.
- You can give a name to each step, as well as a description of what that step does.

Editing exploded view steps

You can access the Exploded View Step Editor in four different ways:

- From the **Edit** menu, choose **Edit Exploded View Steps**.
- In the Design Explorer, right click on the Exploded View and choose **Edit Exploded View Steps**.
- Right-click in the work area, and choose **Edit Exploded View Steps**.
- From the Assembly Modeling toolbar, select the Open Exploded View Step Editor tool

You must be currently viewing an existing exploded view that has at least one part moved from its default location for any of the above options to be available.

Once you have the Exploded View Step Editor open, you can click on each step, and the part associated with that step will be highlighted in the work area. You can name a step, add a description, or change the sequence of the steps.

➢ ***To modify a step:***

1. Click on the step you want to edit.

2. In the **Name** field, type a name that you want to use to refer to that step.

3. In the **Description** field, enter a description of what that step does.

4. Using the movement buttons on the right, move the step up or down in the sequence, or move it to the first or the last position in the sequence.

5. If you would like to combine 2 or more steps into one, press and hold the Control key as you select all of the steps that you want to combine, and then click the **Group Steps** button. (This action can only be reversed by using the Edit - Undo command while viewing the Exploded view, prior to exiting the model.) This capability allows you to show movement of multiple parts simultaneously when the exploded view is published to PDF. (Full Publishing licenses only)

6. Choose **Apply**, then **Close**.

11.8 Deleting an Exploded View

➢ ***To delete an exploded view:***

1. Make sure you are not currently editing any Exploded views.

2. In the Design Explorer, right-click the Exploded view you want to delete and select **Delete** from the pop-up menu; or select the Exploded view in the Design Explorer, and press **Delete** on the keyboard.

11.9 Duplicating an Exploded View

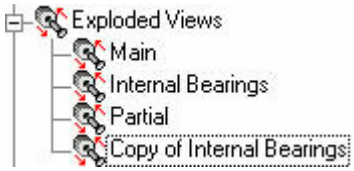
You can create a copy of an existing exploded view.

➢ ***To duplicate an existing exploded view:***

1. Make sure you are not currently editing any Exploded views.

2. In the Design Explorer, right-click the Exploded view you want to duplicate and select **Duplicate** from the pop-up menu; or select the Exploded view in the Design Explorer, and from the **Insert** menu, select **Duplicate Exploded View**.

   A copy of the Exploded view is listed in the Design Explorer.



11.10 Editing and Designing Parts in the Assembly

You can edit existing parts, as well as design new parts, in the context of the assembly. You can reference existing geometry while editing existing parts and designing new parts.

11.10.1 Creating a New Part within an Assembly

➢ ***To create a new part in the assembly:***

1. Click the Insert New Part tool on the Assembly Modeling toolbar.

   Or

   From the **Insert** menu select **New Part** or **New Sheet Metal Part**; or press **Ctrl + Shift + N** (for new part) on the keyboard. The work area changes from assembly edit mode to part edit mode. The Sketching and Part Modeling toolbars are displayed on the right side instead of the Assembly Modeling toolbar. All other parts in the assembly will be displayed semi-transparently.

Note: New Sheet Metal Part is only available in Alibre Design Professional and Alibre Design Expert.

Additionally, the new part is listed in blue in the Design Explorer to signify that it is currently being edited.

2. Construct the part features using the same techniques used to create a part in a part workspace.

   You can use edges and faces on other parts as the sketch plane in the new part. You can also use edges and faces on other parts as references or targets as you model the new part. You can also project entities from existing assembly parts onto a sketch plane in the new part to create sketch or reference figures.

   While in part edit mode, if you use a face on an existing part as a sketch plane or to create a new reference plane, an **inter-design constraint** will automatically be created. See *Inter-Design Constraints* (on page 218) for more information.

3. You can switch back to assembly edit mode at any point. To do so, right-click the part listed in blue in the Design Explorer and select **Edit Root Assembly** from the pop-up menu; or right-click the top-level assembly in the Design Explorer and select **Edit Part/Subassembly** from the pop-up menu.

11.10.2 Editing a Part in an Assembly

You can edit a part without leaving the context of the assembly. While editing a part in assembly mode, you can reference geometry on other parts while sketching or creating new features.

You can also edit a part in an independent workspace as well. After saving and closing the workspace, the changes made to the part will automatically be reflected in the assembly.

➢ ***To edit a part in the context of the assembly:***

1. Right-click on any part in the work area and select **Edit Here**.

   Or

   Right-click the part in the Design Explorer and select **Edit Here** from the pop-up menu.

   The work area changes from assembly edit mode to part edit mode. The Sketching and Part Modeling toolbars are displayed on the right side of the work area instead of the Assembly Modeling toolbar. All other parts in the assembly remain visible but are displayed semi-transparently. The part you are editing remains fully shaded.

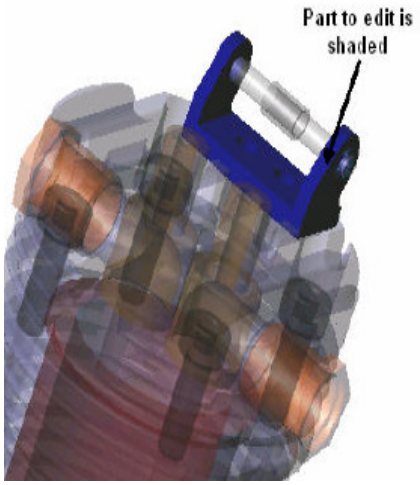
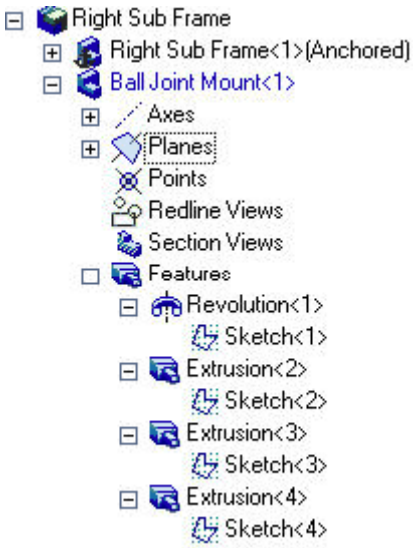


Figure 90: Editing Part in Assembly - Displaying Part Shaded

The part being edited is listed in blue in the Design Explorer and all the associated reference geometry, sketches, and features can be accessed.



2. Edit the part sketches or features just like you would in a part workspace. You can also add new features if required.

3. After making the necessary changes, return to assembly edit mode by right-clicking the part being edited in the Design Explorer and select **Edit Root Assembly**; or right-click the top-level assembly in the Design Explorer and select **Edit Part/Subassembly**.

➢ ***To edit a part in a separate window:***

1. Right-click the part you want to edit in the Design Explorer, or in the work area, and select **Edit in Separate Window**. The part opens in a separate workspace.

2. Make any necessary changes to the part. The changes will be applied to the assembly automatically.

3. When finished editing, close the part workspace. In this case, you do not save the part from the part workspace. You are returned to the assembly workspace and can continue working. When you save the assembly, any parts that you modified will also be saved.

11.10.3 Improving Assembly Performance when Editing Parts

When editing a part in the context of an assembly, you may notice that editing is a little slower than if you were editing it as a stand-alone part (outside the context of an assembly).

The reason for this is that, if your part has inter-part relationships, the assembly must re-evaluate these relationships each time you make an edit (the edits could be any modification, including adding dimensions to a sketch, creating a new sketch, creating a new feature, etc.) The more inter-part relationships you have, the more time is required to evaluate the model.

Inter-part relationships are dependencies that one part has on other parts. For example, projecting the edge of a feature (with Maintain Associativity checked on) into another part to use it to create a feature in the second part. Another example would be creating a sketch and creating a dimension to a feature in another part.

You can control whether or not Alibre Design evaluates all of these relationships on-the-fly, or all at once when you are ready for it to happen. You do this using the Auto-Regenerate tool.

If Auto-Regenerate is checked ON (this is the default behavior), all relationships will be evaluated on-the-fly. If Auto-Regenerate is checked OFF, relationships will be evaluated at a specific time, as determined by one of the situations listed here:

- At any point during edit of the part, you turn ON Auto-Regenerate. The immediate effect of turning ON this switch is that all relationships are evaluated and all pending changes take effect and get displayed on the screen. From this point, all new edits will result in on-the-fly evaluation.
- If you do NOT explicitly turn on Auto-Regenerate, but you exit edit part mode, then at that time all the pending evaluations will be performed and the assembly will be brought up-to-date and its display updated. As a result, exiting edit mode could take a little longer to complete. Auto-Regenerate will continue to remain turned OFF for subsequent editing operations.

➢ ***To turn Auto-Regenerate On and Off:***

From the Tools menu, select Auto-Regenerate. This is an on/off toggle. If the option is checked, it is on. If not checked, it is off.

11.11 Importing Parts into an Assembly

You can import parts and subassemblies into an assembly and subsequently use them in the design. You can import **STEP**, **SAT**, and **IGES** files into an assembly. To learn more about importing data, refer to *Importing and Exporting Data* (on page 305).

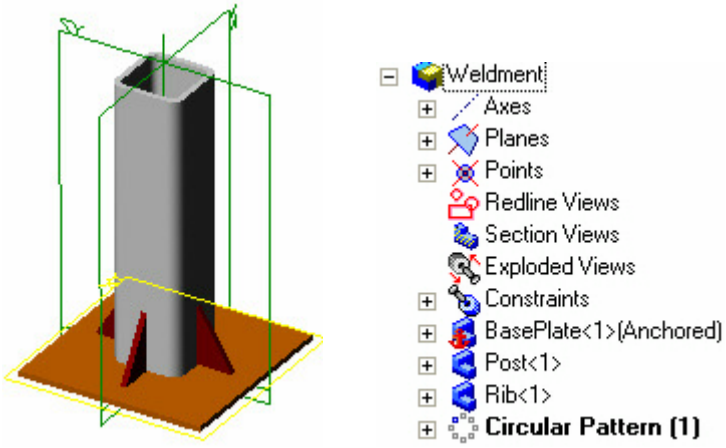
➢ ***To import a part into an assembly:***

1. Select the **Import** tool from the Standard toolbar; or from the **File** menu select **Import**. The **Import File** dialog appears.
2. Browse to the location of the file that you want to import.
3. Select the file to import.
4. Click **Open**. The part/subassembly appears in the assembly workspace and is listed in the Design Explorer. You can constrain the imported parts to existing parts and vice versa.

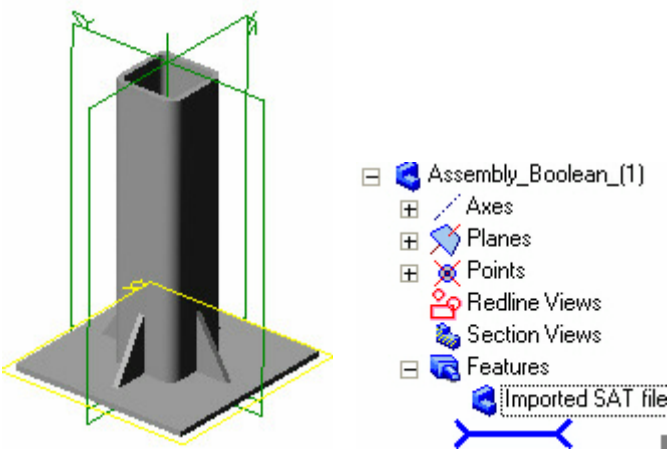
11.12 Joining Parts & Removing Material in an Assembly

You can use the **Assembly Boolean** command to either join multiple parts together into a new single part or to remove material from parts in an assembly.The result is one or more new parts that can be added into the existing assembly or placed in a new workspace.

➢ ***To join parts in an assembly:***



1. From the assembly workspace, select the **Assembly Boolean** command from the **Tools** main menu. The **Assembly Boolean** dialog appears.
2. Select the parts to join from the Design Explorer or the 3D work area. These will be listed in the **Blanks** field.
3. Check the **Join All Blanks** option.
4. If you want the resulting joined part to be placed in the original assembly, check the option **Insert results into current assembly**. If you want the joined part in a separate workspace, uncheck this option.
5. Keep the **Tools** field empty. Click **OK**. The selected parts are joined together. The joined part is represented in the Design Explorer as a new part with an **Imported SAT File** feature.



➢ ***To remove material in an assembly:***

You remove material from parts in the assembly by specifying one or more parts that will be used as tools for cutting away material. From the assembly workspace, select the Assembly Boolean command from the Tools main menu. The Assembly Boolean dialog appears.

1. Select the parts from which you want to remove material from the Design Explorer or the 3D work area. These will be listed in the **Blanks** field.

2. Click in the **Tools** field, and then select the parts you want to use for cutting.

3. Use the **Join All Blanks** option if you want to unite all the blanks into a single part.

4. If you want the resulting cut parts to be placed in the original assembly, check the option **Insert results into current assembly**. If you want the cut parts in separate workspaces, uncheck this option.

5. Click **OK.** The selected blank parts are cut by the selected tool parts. Each cut blank is represented in the Design Explorer as a new part with an **Imported SAT File** feature.

CHAPTER 12

Drawings

You can create 2D drawings of the parts and assemblies you create. Standard 2D views can be created automatically from the part or assembly. Custom views can be created based on the standard views already present in the drawing.

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Dimensioning 250
Inserting Additional Views 256
Custom Templates 265
Annotations 269

12.1 Creating a New Drawing

12.1.1 Opening a New Drawing

You can open a new drawing from the Home window or any open workspace.

➢ ***To open a new drawing from the Home window or any workspace:***

Select the New Drawing tool from Standard toolbar; or from the **File** menu, select **New > Drawing**. The **New Sheet Properties** dialog appears.

12.1.2 Selecting a Drawing Template

You can select from a number of standard drawing templates, your own custom templates, or a blank drawing of varying size. **ANSI**, **DIN**, and **ISO** drawing templates are supported.

➢ ***To select a drawing template:***

1. In the **New Sheet Properties** dialog, select **Template** or **Blank Sheet**.

2. If you selected **Template**, also select a standard drawing template from the list. To use a custom drawing template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog.

   Or,

   If you selected **Blank Sheet**, select a sheet size from the list.

   Note: If you have previously **browsed** to another template folder, you can use the **Default** button to reset the template list back to the system template folder.

3. In the **Default View Scale** area, specify the scale to use in the drawing. The scale can be changed later if necessary.

4. To create a drawing with no models associated with it, check the **Create An Empty Drawing** box.

5. Click **OK**. If you selected a standard or custom template, the **Fill In Text** dialog may appear.

12.1.3 Selecting the Model

You can create 2D drawing views automatically using the views from a part or assembly.

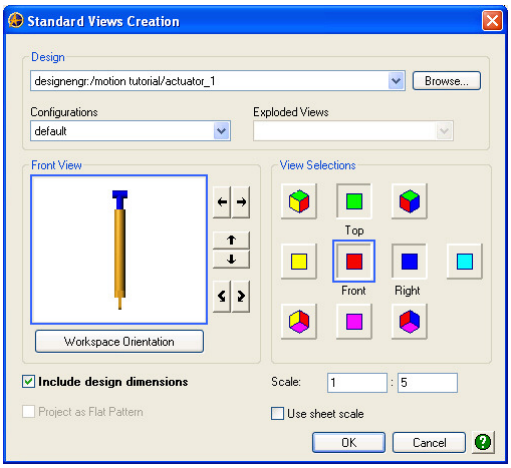
➢ ***To select the model:***

1. In the **Insert Design** dialog, use the Document Browser to select the part or assembly from which the 2D drawing views will be created.

2. Click **OK**. The **Standard Views Creation** dialog appears and the selected design populates the **Design** field.

12.1.4 Inserting Standard Views

You can select which standard views you initially want to create in the drawing. You can insert additional standard views later as necessary.



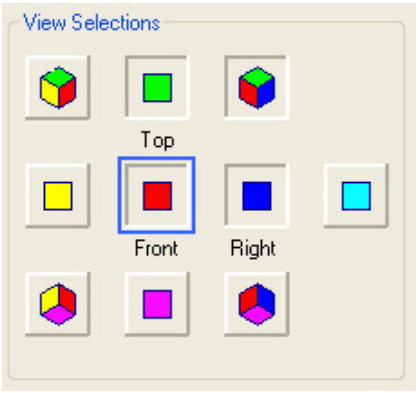
➢ ***To insert the standard views:***

1. A preview of the selected part or assembly is shown in the **Front View** preview window. To select a different part, click the **Browse** button to make a new selection.

2. Select the view that you want to use as the **Front View** in the drawing. You can use the arrow buttons to reorient the view in the preview window.

You can also click the **Workspace Orientation** button to select from default and custom views created in the part or assembly workspace.

3. In the **View Selections** area select the views that you want to insert into the drawing. The default views are **Front**, **Top**, and **Right**. To add or remove a view, click the view's corresponding button.



4. If multiple configurations of the design exist, select the one you want to use for these views from the **Configurations** list.

5. In the case of assemblies, if any exploded views were created in the assembly workspace, you can also select an exploded view from the **Exploded Views** list.

6. To see all of the options at the bottom of the dialog, if they are not visible, click the **More Options** button.

7. In the **View Creation Options** section:

   a. Check the **Fast Views** box to use Fast View Mode on these views. Fast View mode produces a less precise image (visually), but is substantially faster for large or very complex assemblies. Refer to *Fast Views* (on page 236) for more information.

   b. Enter a value for the view scale into the **Scale** field; or, check the option **Use Sheet Scale**, which will select the Sheet scale that was set in the New Sheet Properties dialog.

   c. Select the **Project as Flat Pattern** option if desired for sheet metal parts.

8. In the View Detailing Options section:

   a. Check **Select All** to use all of the available options.

   b. Check **Hidden Lines** to display the entities that would not be directly visible.

   c. Check **Centermarks** and/or **Centerlines** if you want to display them.

   d. Check the **Design Dimensions** box if you want to display the driving dimensions from the model. (These will be the original model dimensions. If you have used any Push/Pull functions, the final model dimensions resulting from those will not be displayed.)

   e. Check **Hole Callouts** to display the full hole callout for holes created using the Hole feature.

   f. Check **Cosmetic Threads** to display a representation of the threads on a threaded hole.

   g. Check **Tangent Edges** to include the tangent edges in the view.

   h. Check **Bend Centerlines** to add center lines to bends in sheet metal parts.

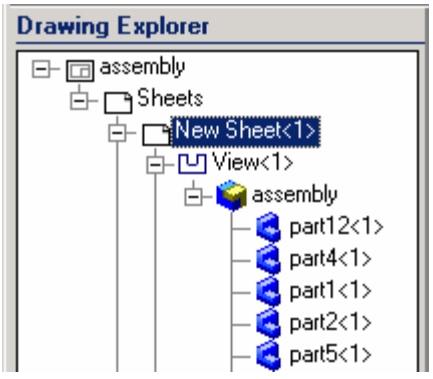
   i. Check **Part Trails** to include the trail lines for exploded views of assemblies.

9. Click **OK** to create the views. Previews of the selected views are displayed in the work area. Note that the cursor is essentially tied to the front view. As you move the cursor, the views will all move together.

10. Move the cursor to dynamically re-position the front view to the correct location within the drawing sheet.

11. Click to place the front view. The corresponding views are also placed. The dimensions that were used to create the part are displayed automatically in the corresponding view (if you checked the option to include design dimensions).

    The **Drawing Explorer** on the left side of the work area lists the sheets associated with the drawing, as well as the views associated with each sheet. The associated design is also listed under the view.



Note: Alibre Design supports both **First Angle** and **Third Angle** projection methods. You can set the projection method for a drawing in the **Detailing** tab of the **Drawing Properties** dialog. Click **Properties** in the **File** main menu to bring up the Drawing Properties dialog.

12.1.5 Inserting Additional Standard Views

After you have created Standard Views, you can create additional standard views that will be scaled and aligned with the initial views you created. For example, you can create a Front and a Right view initially, and then create a Left view later.

➢ ***To create additional standard views***

1. Find the Principal View of the view set. This is the Front View. It is represented in the Design Explorer by the icon.

2. Right click on this view in the Drawing Explorer or directly within the drawing workspace. Select **Other Standard Views...**

3. Select the views you want to add by pressing the buttons that represent them. Views marked with a green check mark already exist in the drawing.

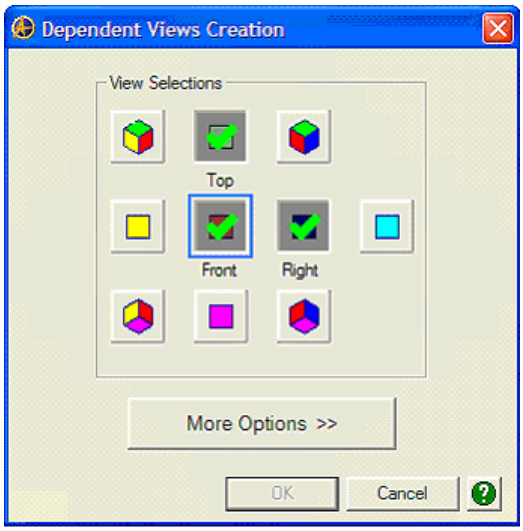


Figure 91: Dependent Views Creation Dialog

4. Select the View Creation options you want for those views using the **More Options...** button. These options are identical to the options in the regular View Creation dialog.

5. Click **OK**. The new view(s) are created and are aligned with and scaled to the Principal View you used during step 2.

Note: You can create as many orthographic (top left, top right, bottom left, bottom right) views as you want. They will never be checked in this dialog.

12.1.6 Fast Views

When you insert views into a drawing, you can choose one of two different modes to create the new views: Fast View or Precise View. Precise View mode is the standard mode and will always be used as the default. Fast Views are useful for large or complex drawings, because they require less time to generate than views in the standard Precise mode. There are some limitations when using Fast Views, so once you are no longer making a large number of changes to the model, you should convert your views to Precise Views. You can include both Fast Views and Precise Views in the same drawing.

Fast Views are noted by a specific icon in the Drawing Explorer , so you can easily tell whether or not a view is in Fast View mode.

➢ ***To use Fast View mode:***

1. Begin the process to insert standard views. See *Inserting Standard Views* (on page 234).

2. In the Standard Views Creation dialog, click the **More Options** button.

3. Check the **Fast Views** option on.

4. Continue selecting the desired options and then click OK to proceed with the view creation.

Benefits of Fast Views

Fast View mode is recommended for use with very large or very complex models. The standard Precise View mode can take a long time to generate drawing views for these models. Using Fast View mode, these views are generated 4-10 times faster, allowing you to get started detailing your drawing much sooner. When you have finished working on your model, and have completed the detailing on the drawing, you can convert your views to Precise Views.

Note: For small models, and models that are not complex, there is no perceptible difference between the two modes, so the standard Precise View mode should be used.

Limitations of Fast Views

The primary difference between Fast Views and Precise Views is that the image created using Fast Views is not as high quality as the image created using Precise View mode. In addition, Fast Views are visually affected by the circular faceting (see Curve Smoothness) value you are using in the 3D workspace. Fast Views should be considered to be "drafts" of the final 2D drawing. You can convert a Fast View to a Precise View at any time to improve the visual quality. You will want to convert your views to Precise Views before printing the 2D drawing, for a higher quality print.

There are certain functions you will not be able to access within a view that was created using Fast Views:

- If an assembly or subassembly contains BOTH inter-design constraints AND design configurations, Fast View mode will not be available when creating drawing views of the assembly.
- You cannot refine edges.
- You cannot set the layers of edges.
- You do not have control over the appearance of hidden or visible lines.
- Fast Views must be reprojected every time you open a drawing which uses them. If the 3D model data (for parts and assemblies) cannot be located, the views cannot be created and will be blank.
- The ability to dimension segment data is not available in Fast Views. The majority of standard 2D drawing detailing practices do not involve using segment data so this will not likely be a problem for most users.

Note: All of these functions become available once the view is converted to a Precise View.

➢ ***To convert Fast Views to Precise Views***

Right click on the view and select **Create Precise View**. If this option is not available, then your view is already a Precise View.

Or,

Right click on the view in the Drawing Explorer and select **Create Precise View**. If this option is not available, then your view is already a Precise View.

12.2 Working in a Drawing

12.2.1 Drawing Mark-Up Mode

Drawing mark-up mode allows you to load a drawing without loading the underlying designs for the drawing. You will enter mark-up mode if the designs are unavailable, or if you have modified any of the designs and choose not to update the drawing. The features available in mark-up mode are all of the features that are accessible immediately after opening a drawing, while the parametric data continues loading, as well as Insert Annotations and Insert Redlines. One benefit to Drawing Mark-up Mode is that you can send another user a drawing to review without sending the design files.

Opening a Drawing that References Missing Items

When you open a drawing that references items that are missing, you have two options: replace the missing item, or delete the views that use the missing items. Immediately upon opening a drawing where this situation occurs, you will see the Missing Designs dialog, shown below. In this case, the last item is missing, as indicated by **[Not Found]**. Missing items are also indicated by their red node color.

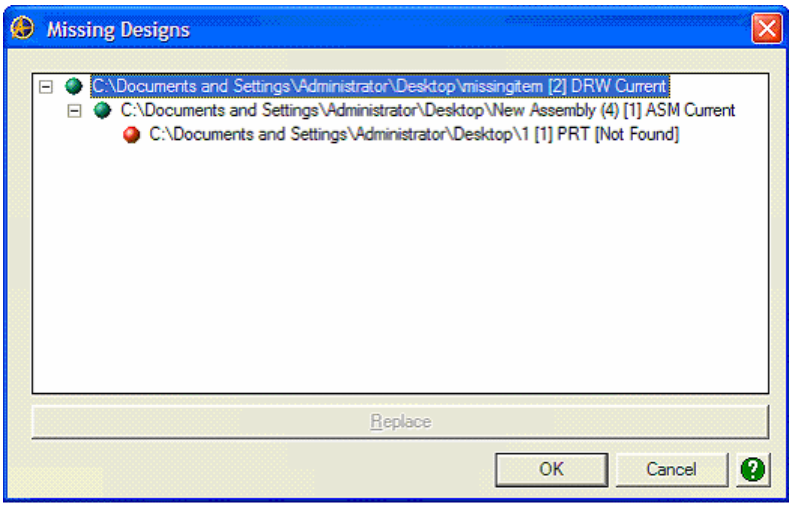


Figure 92: Missing Designs Dialog

➢ ***To replace the missing item with an existing item***

1. Select the item that you want to update. The **Replace** button becomes available.
2. Click **Replace**. The Open dialog appears.
3. Select the item with which you want to replace the missing item.
4. Click **OK**. The description of the item in the Missing Designs dialog changes from **[Not Found]** to **[Replaced]**.
5. Continue the process until all missing items are resolved.
6. Click **OK** to finalize the replacements and open the drawing.

Note: If you replace parts that are part of an assembly in your drawing, saving the drawing will also cause the assembly file to be overwritten, using the new parts in place of the missing ones.

➢ ***To delete views that reference outdated items:***

If you do not want to replace missing items in your drawing, you will have to work in Markup Mode until all views that reference missing items are deleted. When you open a drawing with views that reference missing items, those views will be shown in red in the Drawing Explorer.

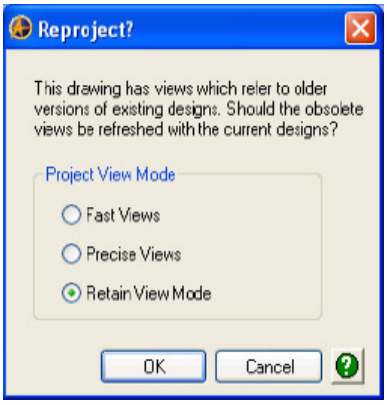
Click each item in red that was indicated to have missing components in the Missing Designs dialog and press **Delete**. Or, right click the items and select **Delete** from the menu.

Once all items that reference missing components are gone, you can continue to fully use the drawing and detailing tools.

Note: Red views in the Drawing Explorer can indicate several things. In general, red views convey that an item is either missing, or that the 3D design has changed and the drawing has not yet been updated to reflect that change. Pay attention to which items are shown in the Missing Design dialog to ensure you do not delete views which simply need to be updated (reprojected) versus views that have missing components.

Opening Outdated Drawings

When you open a drawing from the file system that has an outdated design, the drawing file will open, and you will be prompted to **Reproject** the outdated designs.


Reproject?
This drawing has views which reler to older versions of existing designs. Should the obsolete views be refreshed with the current designs?
Project View Mode
Fast Views
Precise Views
Retain View Mode
OK
Cancel

Choose one of the following options, then click **OK** to continue updating the drawing:

- **Fast Views** - the outdated items will be projected in Fast View mode
- **Precise Views** - the outdated items will be projected in the standard mode
- **Retain View Mode** - the outdated views will be projected in their current mode

Note: If you do not want to reproject your views, click Cancel. If you do not reproject your views, your 2D drawing will continue to be out of date, and you will have limited editing options when working with your 2D drawing until you reproject all outdated views.

12.2.2 Renaming Sheets & Views

You can rename the sheet and corresponding views to convey relevant design information.

➢ ***To rename a sheet or view:***

1. Right-click a sheet or view in the Drawing Explorer and select **Rename** from the pop-up menu.
2. Type the new name.
3. Press **Enter**. The sheet or view name is updated.

12.2.3 Changing the Drawing Template

You can change the drawing template after the initial drawing creation.

➢ ***To change the drawing template:***

1. Right-click the sheet in the Drawing Explorer and select **Change Template** from the pop-up menu. The **New Sheet Properties** dialog appears.
2. Select the **Template** option and select a template from the list. To use a custom drawing template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog.

   Note: If you have previously **browsed** to another template folder, you can use the **Default** button to reset the template list back to the system template folder.

   Or

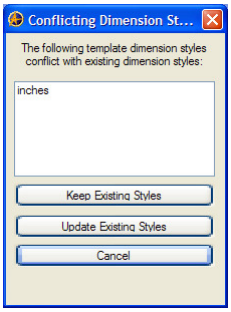
   Select the **Blank Sheet** template option and select a sheet size from the drop down list.
3. Change the **Default View Scale**, if necessary.

4. Click **OK**.

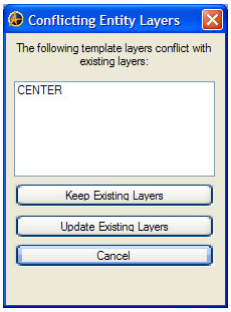
5. If a Standard template was selected, the **Fill In Text** dialog appears. Complete the applicable fields and click **OK**. The sheet is updated and the new template is displayed in the work area.

Note: If the new template has dimension styles or layer properties that conflict with the existing template, you will see the Conflicting Dimension Styles or Conflicting Entity Layers Dialog.

Conflicting Dimension Styles



Conflicting Entity Layers



12.2.4 Deleting Views

You can delete views at any time.

➢ ***To delete a view:***

Right-click the view in the Drawing Explorer and select **Delete** from the pop-up menu; or select a view in the work area or Drawing Explorer and press **Delete**; or select the **Delete** tool from the Standard toolbar.

Note: Any views that were created from the deleted view, such as auxiliary views, will also be deleted.

12.2.5 Hiding Views

You can hide/unhide views at any time. A hidden view is not displayed in the 2D work area. It is displayed in the Drawing Explorer but is grayed out.

➢ ***To hide a view:***

Right-click the view in the Drawing Explorer or the 2D work area and select **Hide** from the pop-up menu.

➢ ***To unhide a view:***

Right-click the view in the Drawing Explorer and select the **Hide** toggle (which should be marked with a check) from the pop-up menu.

12.2.6 Drawing Selection Filters

By default, you can select any item in a drawing. When you move the cursor over an item in the work area, the item is highlighted. You can select the following items individually in a drawing workspace:

- Parts
- Part edges and vertices
- Dimensions
- Sketches
- Annotations
- Redlines
- Views

As you work in a drawing workspace, you may find it advantageous to select a certain group or groups of items as opposed to all items. In this case, you can apply selection filters and specify which item groups you want to select.

➢ ***To use selection filters:***

On the Filters toolbar (see below), select the tools corresponding to the item groups you want to be able to select. A filter is applied when the corresponding tool is in the pressed state. The following filters are available (in order as they appear below):

- Select Segments (drop down list includes Select Parts and Select Views)
- Select 2D Sketches
- Select Dimensions
- Select Annotations
- Select Vertices
- Select Redlines

Only items in the selected groups can be selected in the work area, so as you move your mouse pointer over a view, only applicable items will be highlighted.

You can also apply filters by going to the Tools menu and choosing Selection Filters. If a check mark is displayed next to a filter, the filter is currently being applied.

12.2.7 Moving Views on the Sheet

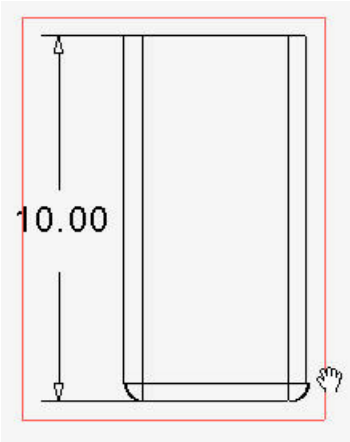
After initially placing the views, you can re-position views on the sheet as necessary. Note that the standard views are initially aligned. Moving the front view will cause related standard views to move. You can break the view alignment if necessary.

➢ ***To move a view:***

1. Select the **View** selection filter if it is not already being applied.

2. Select the **Select** tool from the View toolbar if it is not already selected.

3. Move the cursor over the view. A red view boundary appears and the mouse pointer changes to resemble a hand.



4. To reposition a view or the views, click and hold the mouse button, and move the cursor. The selected view and all associated views move.

5. Release the mouse button to place the views.

Note: You can break an individual view's dependence on other views by right-clicking the view and unchecking the **Align** toggle in the pop-up menu. You can again establish the alignment by rechecking the **Align** toggle.

12.2.8 Updating Drawing Views

It is likely that you will need to make changes to your part and assembly models even after you have created 2D drawings of the models. When you make a change to a part or assembly model and save the model, the 2D drawing becomes outdated, because it does not contain the most recent model information. You can update the drawing to bring in the new model data.

➢ ***To update drawing views:***

1. Select the **Reproject Outdated Views** tool from the Detailing toolbar; or, from the Tools menu, select **Reproject Outdated Views**. The Reproject dialog appears.

2. Choose one of the following options:

   a. **Fast Views** - the outdated items will be projected in Fast View mode

   b. **Precise Views** - the outdated items will be projected in the standard mode

   c. **Retain View Mode** - the outdated views will be projected in whichever mode they are currently in

3. Click **OK** to continue updating the drawing.

12.2.9 View and Sheet Boundaries

A red view boundary is highlighted as you move the cursor over a view. The view boundary is displayed both in and out of sketch mode. The boundary size is calculated automatically based on the extents of the view. Consequently, you cannot change the size of the view boundary.

You can only work on a view (e.g. add dimensions, sketch figures, etc.) when the view boundary is displayed around it. The view boundary indicates the view is active. Any items added to the view when it is active will be associated with the view. If the view is moved, the inserted items will move with it.

A sheet boundary is also highlighted when a sheet is selected and you enter sketch mode. The sheet boundary indicates the sheet is active. Any items added to the sheet when it is active will be associated with the sheet. The boundary size is based on the extents of the entire sheet.

12.2.10 Changing the View Scale

You can change the default view scale or the scale of individual views after they have been placed. A changed default view scale is applied to any views that are inserted subsequent to the scale change. Views already in the sheet will not be effected. Changing the scale of an individual view will subsequently change the scale of its dependent views.

➢ ***To change the default view scale:***

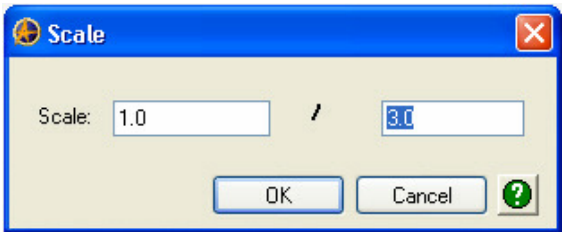
1. In the Drawing Explorer, right-click the sheet in which you want to change the default view scale, and select **Default View Scale** from the pop-up menu. The Scale dialog appears.

2. Modify the scale as required.

3. Click **OK**. The default view scale is changed.

➢ ***To change the scale of a view:***

1. Right-click the view and select **Scale** from the pop-up menu. The **Scale** dialog appears.



2. Specify the new scale.

3. Click **OK**. The view scale is updated.

12.2.11 Line Display in Views

You can control how individual views are displayed. You can show or hide **hidden lines** and **tangent edges**. By default, tangent edges are shown, and hidden lines are not displayed.

➢ ***To show or hide hidden or tangent edge lines in a view:***

Right-click a view in the Drawing Explorer, or right-click a view in the work area and select one or both of the following: **Show Hidden Lines**, or **Show Tangent Edges**. A check mark displayed next to the option indicates the item is currently being displayed.

12.2.12 Centerlines and Centermarks

The following apply to centermarks and centerlines in 2D drawings:

- Displayed by default for holes in 2D drawings.
- Can be modified globally or on an individual basis.
- Dimensions can be placed between centermarks/centerlines and any other applicable item in a view.
- Can be individually deleted, edited, or placed on a different layer.
- Can be inserted on a per view basis.
- Can be inserted individually for projected circular or cylindrical geometry.

➢ ***To turn off automatic centermark and centerline display:***

1. From the **File menu**, select **Properties**. The **Drawing Properties** dialog appears.

2. Click the **Detailing** tab.

3. Under **View Creation Options**, uncheck **Centerlines** and/or **Centermarks**.

4. Click **Apply** and **Close**.

➢ ***To modify global centermark and centerline properties:***

1. From the **File** menu, select **Properties**. The **Drawing Properties** dialog appears.
2. Click the **Detailing** tab.
3. Under **Centerlines**, modify the **Centermark Style**, **Short Dash**, **Extension**, and **Gap** as needed.
4. Click **Apply** and **Close**.

➢ ***To modify individual centermark and centerline properties:***

1. Select the **Select** tool from the **Viewing** toolbar.
2. Move the cursor over the centermark or centerline to be modified.
3. Right-click the centermark or centerline and select **Edit** from the pop up menu. The **Centerline Properties** dialog appears.
4. Modify the **Centermark Style**, **Short Dash**, **Extension**, and **Gap**.
5. If a centermark is being edited, you can rotate it by specifying a **Direction** angle.
6. Click **OK** to accept the changes.

➢ ***To delete a centermark or centerline:***

1. Select the **Select** tool from the **Viewing** toolbar.
2. Move the cursor over the centermark or centerline to be deleted.
3. Right-click the centermark or centerline and select **Delete** from the pop up menu.

   OR

   Right-click the figure and select **Remove Center** from the pop up menu.

➢ ***To insert a centermark or centerline on a per view basis:***

1. Select the **Select** tool from the **Viewing** toolbar.
2. Move the cursor over the view in which centermarks and centerlines are to be added.
3. Right-click the view and select **Insert Centers**.

Note: To insert the centerlines for individual holes, select the hole; then right-click and choose **Insert Center**.

➢ ***To insert a centerline on projected circular or cylindrical geometry:***

1. Select the **Select** tool from the **Viewing** toolbar.
2. Move the cursor over the figure on which to insert a centerline.
3. Right-click the figure and select **Insert Center** from the pop up menu. The centerline is displayed on the figure.

Note: You can also choose to show centermarks and/or centerlines during the view creation process. In the Standard Views Creation dialog, click the **More Options** button to see the View Creation Options. Check the option(s) you want to display in the view. Refer to *Inserting Standard Views* (on page 234) for more information.

12.2.13 Optimizing the Drawing Display

Optimizing your drawing display allows Alibre Design to choose how many segments to break non-linear edges into when projecting them on the display. Smaller edges will be broken down into less segments. This optimization applies to all views in the drawing, and is valid for only projected edges, not for sketches created in the drawing itself.

Turning on the Optimize option reduces the memory used by the drawing and also the time required to render it on the display.

➢ ***To optimize your drawing display***

1. From the **File** menu, choose **Properties**.

2. Select the **Display** tab.

3. In the Curve Smoothness section, check the **Optimize** option to optimize drawing performance.

Optimized edges may appear a little more coarse, because of the smaller number of segments used to display the edge. If you have the optimize option turned on, you can refine individual non-linear edges so that they appear finer.

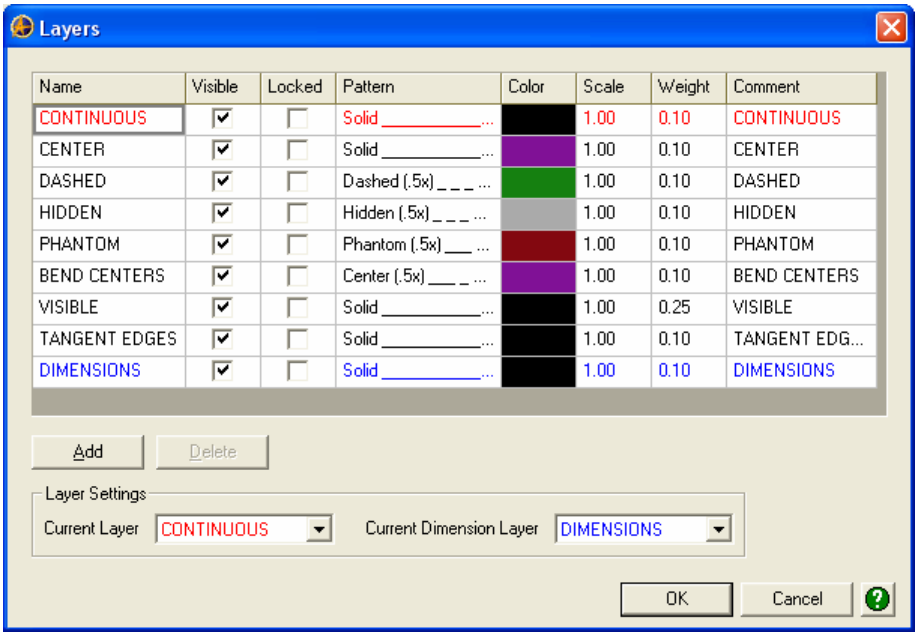
➢ ***To refine individual edges***

Right-click on the edge you want to refine and select **Refine Edge**.

This option will be disabled if the edge has already been refined, or if it is a linear edge.

12.2.14 Layers

In drawings, model views are displayed and detailed using a variety of layers. The use of different layers is often dependant on the drafting and detailing standards defined by your organization. In Alibre Design, there are several predefined layers which can be modified to meet your standards. Additional layers can be added to your pallete as needed. The layer settings are accessible from the Layers tab in the Drawing Properties dialog (**File** > **Properties** > **Layers** tab).



The layer attributes are described as follows:

- **Current** [checkbox]: Layer to be used for new drawing items. Only one layer can be designated as current.
- **Name**: The layer name.
- **Visible** [checkbox]: When checked, all drawing items assigned to that layer are visible. Clear the checkbox to hide items in a particular layer.

- **Locked** [checkbox]: When checked, the corresponding layer is locked and changes cannot be made to layer attributes.
- **Pattern**: A preview of the layer line style. Click to access a menu of additional styles.
- **Color**: The layer color in the drawing. Double-click the colored box to access the Color dialog and select a different color.
- **Scale**: Maximum length of line segments in dashed lines.
- **Comment**: Insert a comment to indicate the layer purpose.

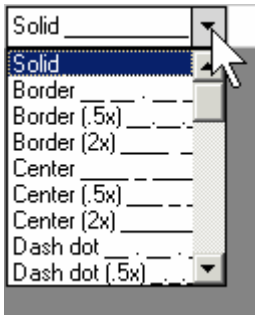
All dimensions will be created on the dimension layer. All other items (e.g., sketch figures, annotations, etc.) will be displayed using the current layer's attributes. If you change the pattern, visibility, color, or scale of a layer used in the drawing, all existing drawing items will be updated with the new settings.

➢ ***To select a different layer as the current layer:***

1. Click the **Current** checkbox corresponding with the layer you want to make current.
2. Click **Apply.**
3. Click **Close.**

➢ ***To change the pattern***

1. Double-click the appropriate Pattern entry. An arrow appears in the cell.



   You can scroll through the list, or type the first letter of the desired pattern name. For example, type the letter "i" to jump to the first ISO style. Continue pressing the "i" key to page through all of the ISO styles.

2. Select the pattern you want from the list.
3. Check **Current** to use this layer for new items.
4. Click **Apply** to implement the new settings.

Note: For advanced users who want to customize the available line patterns in Alibre Design: You can modify the predefined line patterns that ship with Alibre Design by editing the text file, alibre_unicode.lin. You can use Notepad to edit this file. A definition of the file format is embedded within the file. This file is located in the folder C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files.

➢ ***To change the color:***

1. Double-click the appropriate Color cell. The Color dialog appears.
2. Select a preset color or click Define Custom Colors to create a specific color.
3. Click **OK** to close the Color dialog.
4. Click **Apply** to implement the new settings.

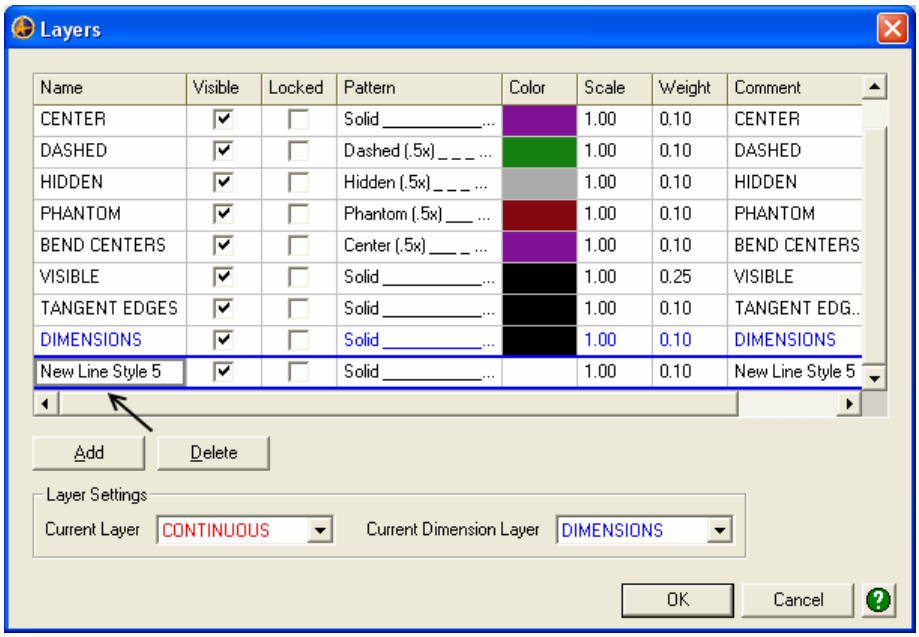
➢ ***To change the layer name, scale, weight, or comment:***

1. Click the cell containing the text or value you want to change. A gray box borders the cell.
2. Click again. A blinking cursor appears. You can now edit the contents of the cell.

3. Click **Apply** to implement the new settings.

➢ ***To add a layer:***

1. Click **Add**. A new row appears at the bottom of the table, temporarily named **New Line Style**.



2. Modify the layer attributes as necessary, including the layer **Name**.

3. Click the **Current** checkbox to use this style for new items.

4. Click **Apply** to implement the changes.

5. Click **Close** when finished.

➢ ***To delete a layer:***

1. Select the layer you want to delete from the list.

2. Click **Delete**.

➢ ***To change the layer of an existing figure, dimension, or annotation:***

1. Right-click the item you want to reformat and select **Set Layer** from the pop-up menu. The **Layers** dialog appears.

2. In the Current column, select the layer to which you want to move the selected item.

3. Click **OK** to apply the change.

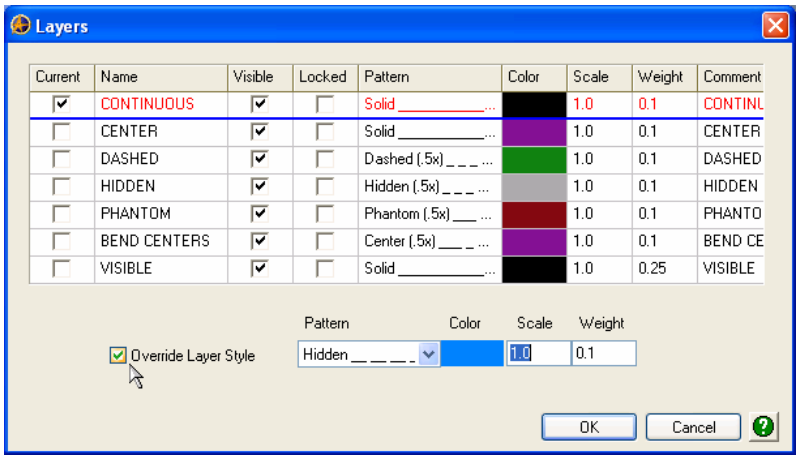
When the layer for an annotation or dimension is changed, the entire dimension/annotation including figures and text will be rendered in the layer's color. All figures and leaders will be rendered with the line pattern of the layer (continuous, dashed, etc.).

The only exception to this rule is the **Text Note** annotation, for which the text is always displayed using the font color specified in the Text Note dialog. The leader will still be rendered in the color and line pattern associated with the layer.

You can change the properties of any figure on a layer without changing the layer on which the figure resides. You do this by overriding the current layer style:

1. Right-click the figure whose properties you want to change.

2. Select **Set Layer**. The Layers dialog appears.



3. Check the **Override Layer Style** box; then select the Pattern, Color, Scale, and Weight for the figure.

4. Click **OK** to apply the changes. The properties of the figure will be changed, but the figure will still reside on the original layer.

12.2.15 Adding Sheets

You can add additional sheets to the drawing as needed.

➢ ***To add a sheet to the drawing:***

1. Select the **Insert New Sheet** tool from the Detailing toolbar; or from the **Insert** menu select **New Sheet**. The **New Sheet Properties** dialog appears.

2. Follow the steps outlined in *Selecting a Drawing Template* (on page 233) and Specifying Standard Drawing Information to specify the information included in the new sheet. After inserting a new sheet into the drawing, the sheet will be listed in the Drawing Explorer under the **Sheets** node.

3. The new sheet will initially be blank. Refer to *Inserting Additional Views* **(on page 256)** for information related to inserting views.

4. To switch between sheets, select a sheet in the Drawing Explorer to view it.

5. To delete a sheet, right-click the sheet in the Drawing Explorer and select **Delete** from the pop-up menu.

12.2.16 Reordering Sheets

You can quickly change the order of the sheets in the drawing file.

➢ ***To reorder the sheets:***

1. Select the **Reorder Sheets** tool from the Detailing toolbar; or, from the **Edit** menu, select Reorder Sheets. The **Reorder Sheets** dialog appears.

2. Click the sheet you want to move.

3. Choose the appropriate button on the right to move the sheet to the desired position.

4. Click **OK** to accept the new sheet order and exit the dialog.

12.2.17 Moving a View to Another Sheet

You can move a view to another sheet in a drawing.

➢ ***To move a view to another sheet:***

1. Right-click the view in the work area or Drawing Explorer and select **Move** from the pop-up menu. The **Select Target Sheet** dialog appears listing the drawing sheets you can move the view to.

2. Select the sheet to which you want to move the view.

3. Click **OK**. The view is moved to the specified sheet and is listed under the target sheet in the Drawing Explorer.

12.2.18 Hiding Parts in a View (Assemblies Only)

In a drawing of an assembly, you can hide parts in a view.

➢ ***To hide a part in a view:***

1. Right-click an edge of the part in the work area view and select **Hide Part** from the pop-up menu.

   OR

   In the Drawing Explorer, click the plus sign ⊞ next to the applicable view.

2. Click the plus sign ⊞ next to the assembly name to expand the list of associated parts.

3. Right-click the part and select **Hide** from the pop-up menu.

4. The part is hidden in the view and is dimmed in the Drawing Explorer. To unhide the part, right-click the hidden part in the Drawing Explorer and deselect **Hide**.

12.2.19 Inserting Images in a Drawing

You can insert images, such as logos, into drawings. Several popular file types are supported: JPG, GIF, TIF, BMP, RLE, DIB, EMF, WMF, PNG, JPE, JPEG, JFIF, and TIFF. Alibre Design does not support JPEG 2000 or lossless JPEGs, as well as some types of TIFF files.

When you save the drawing, the images are compressed and stored with the drawing. Opening the drawing opens all the images inside it.

When you export a drawing in the Alibre Design STEP format, the images are compressed and saved with it. When the STEP file is opened, the images are decompressed and loaded into the file.

When you save a drawing as a template after inserting an image, the image will remain part of the template when it is used again in another drawing.

➢ ***To insert an image in a drawing:***

1. From the **Insert** menu, select **Image**. The Select Image dialog appears.
2. Click **Look In** to browse to the saved location of the image you want to insert.
3. Click the image's file name.
4. Click **Open**. The image appears in the drawing workspace and the cursor changes to a dotted crosshairs icon.
5. Click the crosshairs icon on the drawing sheet to place the upper-left-hand corner of the image. The image is placed in the selected location.
6. To resize the image, right-click it and select **Scale** from the pop-up menu. The Scale dialog appears.
7. To make the image smaller, decrease the value on the left. To make the image larger, decrease the value on the right.
8. Click and drag the image to place it as necessary.

Note: You may also move images into Alibre Design by dragging them from a Windows folder onto the Home window. A blank drawing sheet opens with the image placed on it.

12.2.20 Printing a Drawing

You can print one, all, or a specified list of sheets in a drawing. You can also print just a portion of the current sheet.

➢ ***To print:***

1. From the File menu, select **Print**.
2. Use the **Sheet range** to specify what you want to print:
    - All sheets in the drawing.
    - The currently displayed portion of the current sheet.
    - The entire current sheet.
    - Only the sheets checked in the print dialog.
3. Specify the number of copies you want to print.
4. If you want the printed drawing to be fit to the size of the paper, check the **Scale to fit** option.
5. If you want to print using only black and white, check the **Print black and white** option.
6. Click **OK**.

➢ ***To see a print preview of the current sheet:***

From the **File** menu, select **Print Preview**. The **Print Preview** window appears with a preview of the drawing.

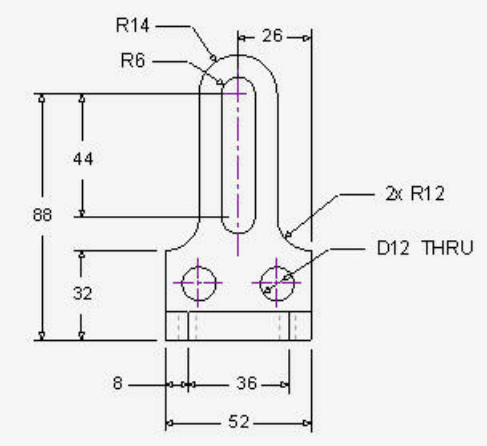
➢ ***To optimize the quality of your print:***

If your drawing contains *Fast Views* (on page 236), you should convert them to Precise Views before printing for better print quality.

12.3 Dimensioning

Typically, as you create features in part mode you place driving dimensions that define the associated sketch profiles. Additionally, dimensions that define a feature's size, such as extrusion depth, are also considered driving dimensions.

When you create a drawing based on the part, you can choose to display the **driving** dimensions from the part automatically during *view creation* (see "Inserting Standard Views" on page 234) on the applicable view in a drawing. If you do not choose to display driving dimensions upon creating the drawing you can always do so later.



You can also manually insert additional dimensions on a drawing view. These user-added dimensions are referred to as **Reference** dimensions. Reference dimensions are displayed the same as driving dimensions. However, you can display a reference dimension in parentheses to distinguish it from driving dimensions. Right-click the reference dimension and select **Properties** from the pop-up menu. Select the **Units and Tolerance** tab, select the **Display As Reference Dimension** option, and click **OK**.

If a driving dimension is changed in part mode, the associated dimension in the drawing will automatically get updated. You can also edit driving dimensions in the drawing. If you change a driving dimension in the drawing, the part is automatically updated.

Reference dimensions cannot be edited or changed. However, reference dimensions will update upon the modification of driving dimensions. You can insert additional reference dimensions in drawings to further clarify the design intent.

12.3.1 Placing Additional Dimensions on a View

You can use the same methods to add dimensions to a drawing as you use to dimension sketches in part mode. Refer to *Sketching* (on page 25) for more information.

➢ ***To place dimensions in a drawing:***

1. Select the **Dimension** tool from the Sketching toolbar.
2. Select the figure to dimension. A dimension preview appears, press **Esc** if the wrong figure was dimensioned.
3. When the preview displays the correct dimension location, click to place the dimension. The newly placed dimension is a reference dimension.

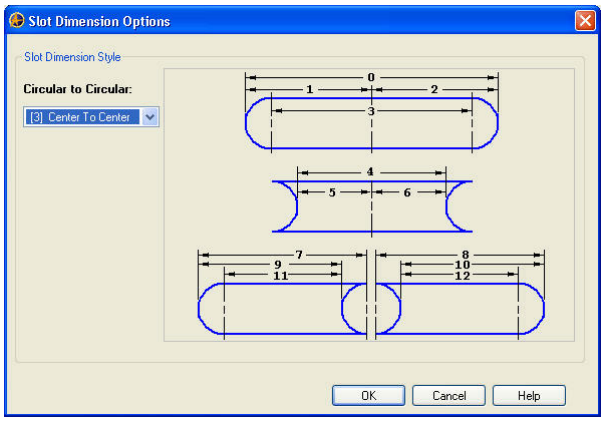
12.3.2 Dimensioning Slots and Holes

You can easily control how dimensions related to slots and holes are created and displayed.

➢ ***To insert a dimension related to a slot or hole:***

1. Select the **Dimension** tool from the Sketching toolbar.
2. Select the first circular/radial figure or line to be dimensioned from. A dimension appears but do not click to place it.

3. Select a second circular/radial figure or line to be dimensioned to. The **Slot Dimension Options** dialog appears.



4. From the **Circular to Circular** or **Linear to Circular** list, select a dimension type from the drop down list. The numbers in the list correspond to the graphical key in the dialog.

5. Click **OK**. The dimension is previewed in the work area.

6. Move the cursor to correctly position the dimension.

7. Click to place the dimension.

12.3.3 Placing Ordinate Dimensions on a View

➢ ***To place ordinate dimensions:***

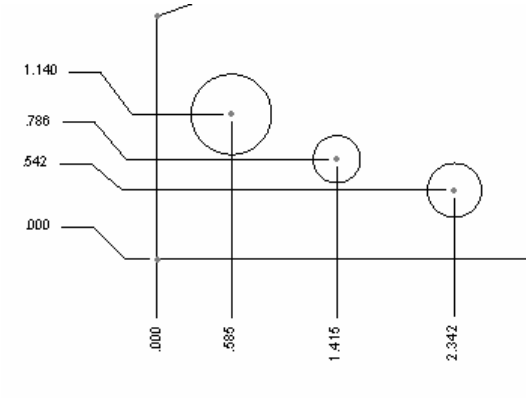
1. Right-click in the work area and select **Ordinate Dimension** from the pop-up menu; or from the **Sketch** menu select **Ordinate Dimension**.

2. Select the **Baseline** figure.

3. Select the **Origin** on the baseline figure.

4. Drag the origin's dimension line away from the model and click to place it.

5. Select the entities (edges or points) you want to dimension using the same ordinate. As you select each entity, the dimension is placed in the view aligned to the origin.



Note: You can insert additional ordinate dimensions to a chain after initial placement. Select the Dimension tool, pick the baseline dimension, and then select the new dimension location to add the new dimension to the chain.

12.3.4 Modifying Driving Dimension Values

As previously described, you can change the values of driving dimensions in drawings. A change to a driving dimension in a drawing will be reflected in the part automatically.

➢ ***To modify a driving dimension:***

1. Make sure the **Dimension** *selection filter* (see "Drawing Selection Filters" on page 241) is on.
2. Select the **Select** tool from the View toolbar if it is not already selected.
3. Move the cursor over the dimension. The dimension turns red.
4. Double-click the dimension. The dimension control box appears.

5. Enter a new value in the box and press **Enter**.

Changes made to driving dimensions in the drawing will be reflected in the part after the drawing has been saved and the part is opened.

12.3.5 Dimension Properties

Dimension properties are determined by the style that you assign to each dimension.

When a dimension is created, it is automatically assigned to the dimension style that you have set as your default dimension style.

You can change dimension properties either as a group or on an individual basis.

➢ ***To change the default dimension style***

1. From the **File** menu, select **Properties**.
2. Select the **Dimensions and Annotations** tab.
3. In the **Dimensions** section, select your desired default style from the drop-down list. This style will be applied to all newly-created dimensions.
4. Click **Apply**, then **Close.**

➢ ***To change the properties of a single dimension***

1. Right-click the dimension and select **Properties**. The Dimension Properties dialog appears.
2. To change which style is used for that dimension, select a new style from the **Dimension Style** drop-down list.
3. To set the properties for this dimension independent of any style, uncheck the **Use Style** checkbox. You can then set the properties as desired for each tab.
4. Click **OK**.

➢ ***To change the properties of multiple dimensions***

1. Click the first dimension to select it, then hold the Shift key down as you select each subsequent dimension.
2. Right-click on any of the selected dimensions and choose **Properties**. The Dimension Properties dialog appears. The values for the FIRST dimension selected will be shown in the dialog. All values that are common between the selected dimensions will be shown with a white background. If any values differ between the selected dimensions, that field will be shown with a slightly gray background.
3. To change which style is used for these dimension, select a new style from the **Dimension Style** drop-down list.

4. To set the properties for this dimension independent of any style, uncheck the **Use Style** checkbox. You can then set the properties as desired for each tab.

5. Click **OK**.

12.3.6 Dimension Styles

Dimension styles are used to set the dimension properties in a 2D detailed drawing. When a dimension is created, it is automatically assigned to the dimension style that you have set as your default dimension style.

You can create multiple dimension styles in a drawing. The styles are saved with the drawing file, not with your system profile, so you will not see the styles you have created in another drawing.

If you anticipate that you will use the same styles again and again, you can save them in your drawing template. Then, each time you use the template, the styles you created will be available.

Setting Your Default Style

All newly inserted dimensions will automatically be assigned to the default style (you can change the style used for an individual dimension after it has been created). You can change the default style used in your drawing by going to **File > Properties**, and choosing the **Dimensions** tab. In **Default Style**, choose the style you want to use from the drop-down list.

- ***To Edit an Existing Dimension Style:***

1. From the **Tools** menu, select **Dimension Styles**. The Dimension Styles dialog appears.

2. Choose the dimension you want to modify from the **Dimension Style** drop-down list.

3. Make any necessary modifications on each of the tabs.

4. Click **OK** to accept the changes and exit the dialog, OR choose another style to modify from the drop-down list, OR choose to create a new style.

- ***To Create a New Dimension Style:***

1. From the **Tools** menu, select **Dimension Styles**. The Dimension Styles dialog appears.

2. Click the **New Dimension Style** button.

3. In **Name**, enter the name for the new style.

4. In **Copy From**, select the existing style you want to use for the initial values.

5. Click **OK**.

6. Make any necessary modifications on each of the tabs.

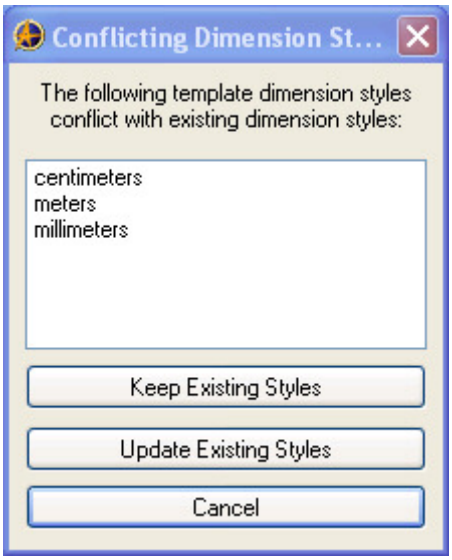
7. Click **OK** to accept the changes and exit the dialog, or choose another style to modify from the drop-down list, or choose to create a new style.

Using Dimension Styles in Templates

Saving your dimension styles to your template allows you to reuse the same styles without having to create them again each time you start a new drawing.

Note: We recommend that you give all of your styles a unique name, even if they are in different templates. This will prevent conflicts from arising if you choose to use more than one template in the same drawing file.

If you do use the same style names for styles in different templates, you may see the following dialog when you attempt to use both templates in the same drawing:



This dialog is telling you that two styles with the same name cannot exist in a single drawing file. The dialog in the above example has found three different styles in the second template brought into a drawing that have the same name as styles in the first template used in the drawing.

You must choose one of the options:

These options only apply to the styles that have the same name. All other styles are unaffected. In addition, these options do not modify the template itself, only the properties of the styles in this drawing file. The original templates remain unchanged.

- **Keep Existing Styles** - This option will keep the properties of the styles that are already in the drawing.
- **Update Existing Styles** - This option will keep the properties of the styles that are in the new template being added, so the existing style properties will be updated to match.
- **Cancel** - This option will cancel the action you just started, whether it was inserting a new sheet, or changing templates.

If you change the template of any sheet in the drawing, the default style for the drawing will update to match the default style of the template you have changed to.

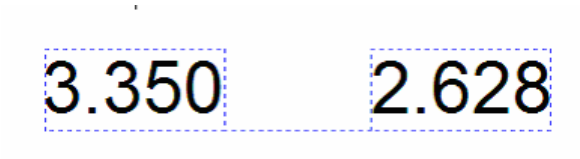
12.3.7 Aligning Dimensions

There is an option to turn on alignment snapping for annotations. This option allows dimensions to snap to other dimensions and annotations to snap to other annotations.

➢ ***To turn on and use dimension alignment snapping:***

1. Select **Tools > Options** in a Drawing workspace.
2. On the General tab, make sure that **Align annotations when dragging** is checked.
3. Exit the Options dialog.
4. Ensure you are not in Sketch Mode.

5. Drag dimensions near other dimensions. You will see inference lines indicating the dimensions are lined up.



Several types of inferencing are available: left edge, right edge, top edge, bottom edge, vertical center, and horizontal center.

12.4 Inserting Additional Views

In addition to the standard views, you can insert additional custom views to further clarify design intent. You can insert **auxiliary**, **detail**, **section**, and **exploded** views.

You can also insert flat pattern views of sheet metal parts in Alibre Design Professional.

Note: When you insert additional views that are dependent views (such as the detail view), the new view will be created with the same projection method as the parent view (Fast View or Precise View). Once the new view has been placed, you can control the projection type independently.

12.4.1 Standard View

You can insert an additional standard view as needed in one of two ways. If you want the new standard views to be aligned with existing views, see *Inserting Additional Standard Views* (on page 236).

If you do not want the new views to be aligned, or if you want the new views to be of a different design, use the steps below.

➢ ***To insert a standard view:***

1. Select the **Standard View** tool from the Detailing toolbar; or from the **Insert** menu select **Standard View**. The **Insert Design** dialog appears.

2. *Select the model* (see "Selecting the Model" on page 233) and choose *which views you want to insert* (see "Inserting Standard Views" on page 234).

12.4.2 Auxiliary View

An auxiliary view is created by projecting an orthogonal view normal to a linear edge or sketch line in an existing view.

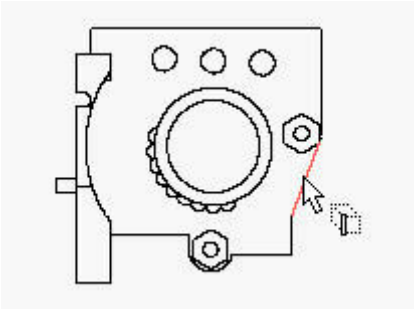
➢ ***To create an auxiliary view:***

1. Select the **Auxiliary View** tool from the Detailing toolbar; or from the **Insert** menu select **Auxiliary View**.

2. Select a linear edge or straight sketch line on an existing view. The edge selected from the parent view must reside on a face that is perpendicular to the plane of the screen.

Note: To use a sketch line as the projection line, the sketch line must reside on the view. To do this, first select the view you want to use to create the auxiliary view. Then select the **Activate 2D Sketch** tool from the sketching toolbar and sketch a line. The line can intersect the view boundaries. Exit Sketch Mode, then follow steps 1 and 2 above to create the new view. (After the auxiliary view is created, the sketch line can be deleted.)

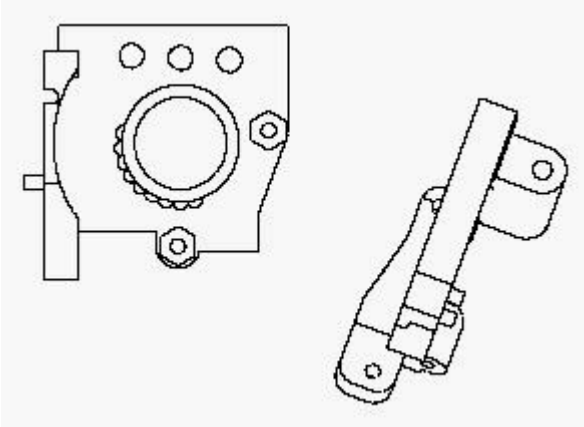
A preview of the auxiliary view is displayed.



3. Move the cursor to position the auxiliary view to the correct location.

4. Click to place the auxiliary view. The auxiliary view is placed on the sheet, aligned to the edge from which it was created. You can only move the auxiliary view in the direction normal to the edge from which it was created.

Note: To break the alignment between the auxiliary view and the parent view, right-click the view that is currently aligned, and select **Align**. Selecting Align will uncheck that option, and you can then drag the view unconstrained.



12.4.3 Detail View

A detail view is a view that shows a portion of an existing view at an enlarged scale.

➢ ***To create a detail view:***

1. Select the **Sketch Mode** tool from the Sketching toolbar.

2. Sketch any closed figure enclosing the area that you want to detail.

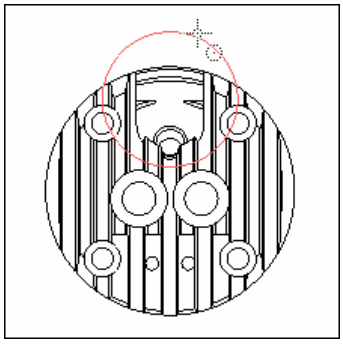


Figure 93: Drawing view with closed shape outlining detail area

3. Select the **Detail View** tool from the Detailing toolbar; or from the **Insert** menu select **Detail View**.

4. Select the sketch figure. A preview of the detail view appears.

5. Move the mouse to position the detail view appropriately. The detail view can be placed anywhere on the sheet. Click to locate the view. The default scale for the detail view is the same as the sheet scale. To enlarge the detail view, right-click it and select **Scale** from the pop-up menu.

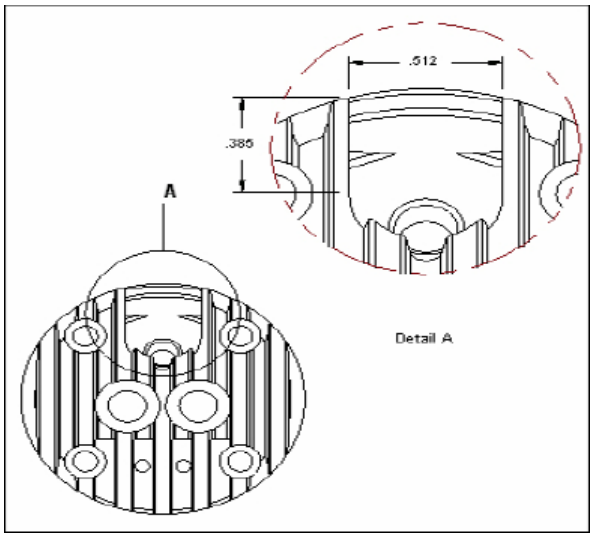


Figure 94: Completed Detail View

You can place dimensions on the detail view. You can also change both the detail circle label and the detail note. To change either, double-click the text or right-click the text and select **Edit** from the pop-up menu. The **Note** dialog appears containing the original text. Enter the new text and click **OK**.

Also, in the **Detailing** tab of the **Drawing Properties** dialog, you can pre-define the border style and font used for detail circles and detail view labels.

When using a sketched circle for the detail view, the leader line and annotation positions are determined by the last location of the mouse before finalizing the circle you sketched as the reference.

➢ ***To change the detail location:***

1. Click the **Select** tool from the View toolbar if it is not already selected.

2. Move the mouse pointer over the detail circle annotation. The annotation is highlighted and the mouse pointer changes.

3. Click and drag the detail circle annotation to the new location.

➢ ***To change the detail area size:***

1. Click the Select tool from the View toolbar if it is not already selected.

2. Move the cursor over the detail circle annotation. The annotation is highlighted and the cursor changes.

3. Double-click the annotation. The **Detail View Annotation** dialog appears and the circle center node is displayed.

4. Move the cursor over the circle.

5. Click and drag to resize.

6. Release the mouse button when the circle is resized appropriately.

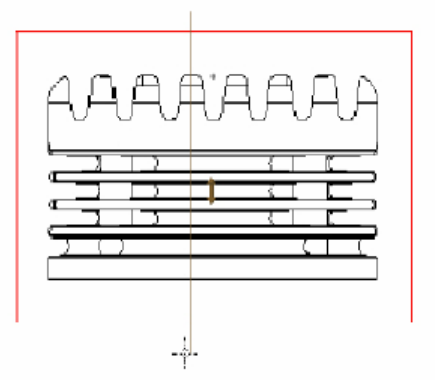
7. Click **OK** in the Detail View Annotation dialog. The detail view is updated automatically to reflect the change in size of the detail area.

12.4.4 Section View

A section view represents a 2D cross-section of a model. Section views are created from other views and are dependent on them. You can create a normal section view or a stepped section view.

➢ ***To create a normal section view:***

1. Sketch a straight line across the view to define the section location. Make sure you are sketching in the view by selecting the view in the Design Explorer before entering Sketch Mode.

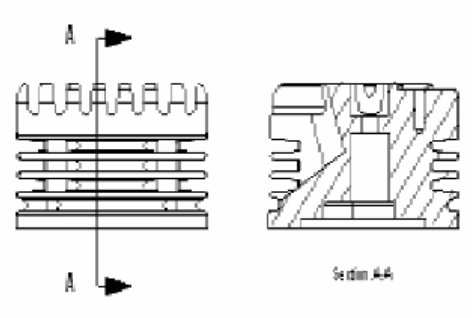


2. Select the **Section View** tool from the Detailing toolbar; or from the **Insert** menu select **Section View**.

3. Select the line that you sketched in step 1. A preview of the section view appears in the drawing.

4. Drag the section view preview to the appropriate location on the sheet. You will only be able to move the view in a direction normal to the section line.

5. Click to place the view.



Notes:

In the **Detailing** tab of the **Drawing Properties** dialog, you can pre-define the display style for the section line, the font for the section line label, and the default hatch pattern style.

For advanced users who want to customize the available hatch patterns in Alibre Design: You can modify the predefined hatch patterns that ship with Alibre Design by editing the text file, alibre_unicode.pat. You can use Notepad to edit this file. A definition of the file format is embedded within the file. This file is located in the folder C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files.

➢ ***To modify a section view:***

- To edit the letter label on the section line or the note on the section view, double-click the item, or right-click and select **Edit** from the pop-up menu. The **Note** dialog appears containing the original text. Enter the new text and click **OK**.
- To change the cut direction, right-click the section view and select **Reverse Section View** from the pop-up menu.
- The section view crosshatch pattern is set in the **Detailing** tab on the **Design Properties** dialog (**File > Properties**). You can modify the crosshatch pattern for an individual section view. Right-click the section view and select **Change Cross Hatch** from the pop-up menu. The **Hatch Properties** dialog appears. Select a new crosshatch pattern from the list or modify the **Scale** or **Angle**. Click **OK** to apply the changes.

➢ ***To create a stepped section view:***

1. Sketch a series of line segments across the view to define the section location.

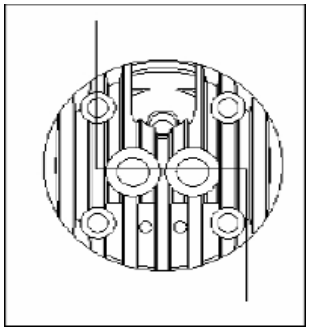


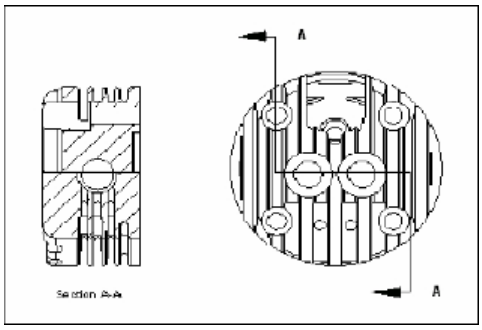
Figure 95: Drawing view showing stepped outline for section view

2. Select the **Section View** tool from the Detailing toolbar; or from the **Insert** menu, select **Section View**.

3. Select any of the line segments that you sketched in step 1. A preview of the section view appears in the drawing.

4. Drag the section view preview to the appropriate location on the sheet. You will only be able to move the view in a direction normal to the section line.

5. Click to place the view.



➢ ***To redefine the section location (either normal or stepped):***

1. Select the Select tool from the View toolbar if it is not already selected.

2. Move the cursor over the section line annotation. The annotation is highlighted and the cursor changes.

3. Click and drag the line to the desired location.

4. Release the mouse button to place the line. The section view updates automatically based on the new section line location.

➢ ***To change the direction of the section view:***

1. Right-click the section view in the work area and select **Reverse Section View** from the pop-up menu.

   Or

   Right-click the section view in the Drawing Explorer and select **Reverse Section View** from the pop-up menu.

2. The section line is flipped and the section view is displayed from the opposite direction.

➢ ***To change the hatch pattern for all parts in the section view:***

1. Right-click the section view in the work area and select **Change Cross Hatch** from the pop-up menu; or right-click the section view in the Drawing Explorer and select **Change Cross Hatch** from the pop-up menu. The **Hatch Properties** dialog appears.
2. Select a new cross hatch pattern from the **Pattern** pull down menu.
3. Specify the cross hatch **Scale**.
4. Specify the cross hatch **Angle**.
5. If desired, you can also modify the **color**, **line weight**, and **offset** distance for the hatch pattern.
6. Click **OK** to apply the new cross hatch pattern.

Note: You can also access the cross hatch settings from the Drawing Properties dialog. From the **File** menu, select **Properties** and then select the **Detailing** tab in the dialog.

➢ ***To change the hatch pattern for individual parts in a section view:***

1. In the Drawing Explorer, expand the assembly under the section view to reveal the individual parts displayed in the section.
2. Right-click the desired part and choose **Change Cross Hatch** from the pop-up menu. The Hatch Properties dialog appears.
3. Modify the hatch pattern properties as desired.
4. Click **OK** to apply the new cross hatch pattern to the selected part.

➢ ***To exclude parts from being sectioned in a section view***

1. Create the section view. All parts will be sectioned at this point.
2. In the section view you just created, right click on any edge of the part you would like to not be sectioned in the section view.
3. Select **Do Not Section**. The section view is recomputed and the part you indicated is no longer sectioned.

Note: Do Not Section is only available when the section view contains more than one part.

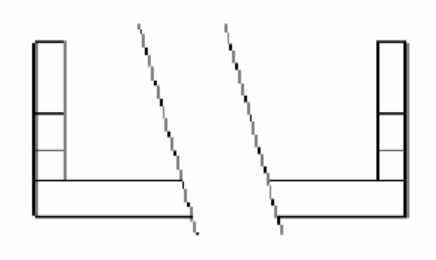
12.4.5 Broken View

You can create a broken view in a drawing of a long part that has a uniform cross-section. Creating a broken view is useful when you want to display a part with a larger scale on a smaller size drawing sheet.

➢ ***To insert a broken view:***

1. Select the Select tool from the View toolbar if it is not already selected.
2. Right-click the view you want to break in the Drawing Explorer or the work area and select **Create Broken View** from the pop-up menu; or select the view and then from the **Insert** menu, select **Broken View**; or select the view and then select the **Broken View** tool from the Detailing toolbar.
3. Two break lines appear in the view and the **Broken View** dialog appears.

4. Select a break line **Style**. You can use **Straight**, **Zig**, or **Curve** break lines.
5. Drag the break lines to the appropriate break locations in the view.
6. Specify the break **Width**.
7. Specify a break line **Angle** if desired.
8. Click **OK**. The part is displayed with a break in the geometry.



Reference dimensions and part dimensions associated with the broken area reflect the actual value.

➢ ***To modify the broken view:***

1. Click the **Select** tool from the View toolbar if it is not already selected.
2. Move the cursor over one of the break lines.
3. Right-click the break line and select **Edit** from the pop-up menu. The **Broken View** dialog appears.
4. Change the **Style**, **Width**, or **Angle** as necessary.
5. Drag the break lines to redefine the break position.
6. Click **OK** when finished.

➢ ***To restore the broken view to it original state:***

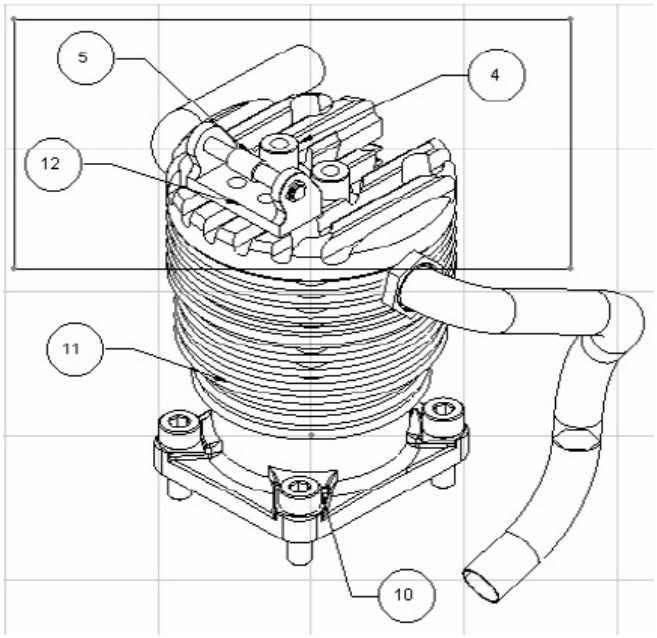
1. Select the **Select** tool from the View toolbar if it is not already selected.
2. Move the cursor over one of the break lines.
3. Right-click the break line and select **Delete** from the pop-up menu. The break lines are deleted and the view is restored to its unbroken state.

12.4.6 Partial View

You can modify an existing view to create a partial view. A partial view allows you to only show a portion of an existing view.

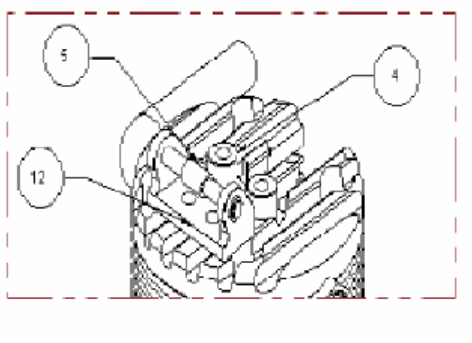
➢ ***To insert a partial view:***

1. Click the Select tool from the View toolbar if it is not already selected.
2. Select the view that you want to transform into a partial view, then select the **Sketch Mode** tool from the Sketching toolbar. Partial views can be created from any other view, including primary and dependent views.
3. Sketch any closed figure enclosing the area that you want to keep in the partial view.



4. Select the **Partial View** tool from the Detailing toolbar; or from the **Insert** menu select **Partial View**.

5. Select the sketched closed figure. The view will transform into a partial view.

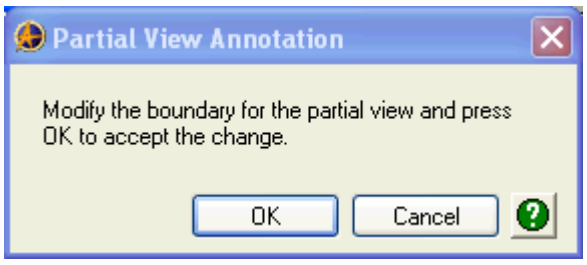


➢ ***To change the partial view area size:***

1. Select the Select tool from the View toolbar if it is not already selected.

2. Move the cursor over the partial view area dashed outline.

3. Right-click the dashed outline and select **Edit**. The **Partial View Annotation** dialog appears.



4. Make any necessary changes to the partial view outline. You can only modify the exiting sketch lines; you cannot sketch new ones.

5. In the Partial View Annotation dialog, select **Cancel** to discard the changes, or **OK** to update the partial view.

➢ ***To move the partial view area:***

1. Click the **Select** tool from the View toolbar if it is not already selected.
2. Move the mouse pointer over the partial view area dashed outline.
3. Click and drag the outline to a new position in the view. You will see a dialog letting you know that moving the annotation will update the dependent views.
4. Choose **Yes** to continue, or **No** to cancel the operation. If you choose Yes, the partial view is updated automatically to reflect the view outline's position change. (You can turn the prompt off if desired - from the **Tools** menu, select **Options**. On the **General** tab, uncheck **Prompt when detail, section, or partial view annotation moved**.)

➢ ***To transform the partial view back to its original state:***

1. Choose the **Select** tool from the View toolbar if it is not already selected.
2. Move the mouse pointer over the partial view area dashed outline.
3. Right-click the dashed outline and select delete.

12.4.7 Exploded View

You can insert a 2D exploded view representation of any exploded view you created in the assembly workspace.

➢ ***To create an exploded view of an assembly:***

1. Select the **Standard Views** tool from the Detailing toolbar; or from the **Insert** menu, select **Standard Views**. The **Standard Views Creation** dialog appears.
2. In the **Design** area, select the assembly item from the drop down list that you want to insert an exploded view of; or click **Browse** to select the assembly from the Document Browser. Any exploded views that were saved with the assembly are subsequently listed in the Exploded Views drop down list.
3. Select the appropriate exploded view to insert.
4. Use the arrow buttons in the **Front View** area, as well as the View Selection buttons to select the appropriate exploded view orientation(s).
5. Click **OK**. A preview of the exploded view appears in the work area.
6. Move the cursor to position the view.
7. Click to place the exploded view on the sheet.

Note: You can toggle on and off the trail lines of an exploded view by right clicking on the view and selecting Show Part Trails.

12.4.8 Flat Pattern View of a Sheet Metal Part

You can insert a flat pattern representation of any sheet metal part you created. Bend lines will be shown in a flat pattern view, and they can be used to create dimensions.

➢ ***To create a flat pattern view of a sheet metal part:***

1. Select the **Standard Views** tool from the Detailing toolbar; or from the **Insert** menu, select **Standard Views**. The Standard Views Creation dialog appears.
2. Follow steps 1-4 for creating standard views, and then click the **More Options** button to see all of the View Creation Options. Refer to *Inserting Standard Views* (on page 234) for more information.

3. Check the **Project as Flat Pattern** box.

4. Click **OK**. A preview of the flat pattern view appears in the work area.

5. Move the cursor to position the view.

6. Click once to place the view on the sheet.

12.5 Custom Templates

You can create a custom drawing template that meets your design process requirements. A custom template can be created in two different ways: as a symbol that you insert into a drawing, or saved as a drawing that you select from the **New Sheet Properties** dialog.

12.5.1 Creating a Custom Template

➢ ***To create a custom template:***

1. Open a new drawing workspace. The **New Sheet Properties** dialog appears.

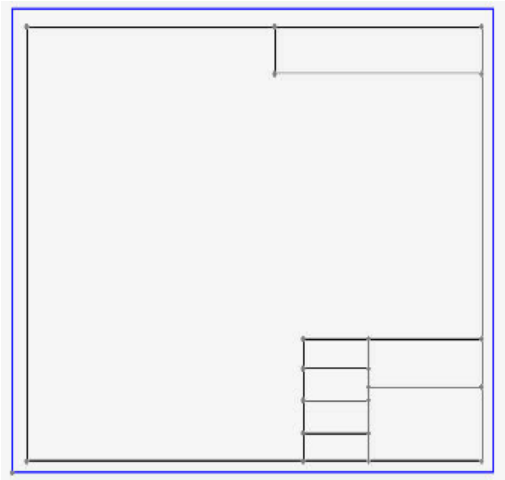
2. Select **Blank Sheet**.

3. Select a sheet size from the pull-down list.

4. Click the **Create An Empty Drawing** checkbox.

5. Click **OK**. An empty drawing workspace appears.

6. In the work area, sketch the border and title block using the sketch tools.



7. Create text labels or standard information such as your company's name and address. From the **Sketch** menu select **Text** > **Label**. Text labels can be used for making title block labels such as **Scale**, **DRW**, **PART NO.**, etc.

8. Set up fields that collect data from the user for variable information such as Drawn by, Date, Designed For, Drawing Number, Scale, etc. From the **Sketch** menu select **Text** > **Field**. Text fields can be used for making title block fields to receive data that corresponds with SCALE, DRW, PART NO., etc. See *Automatically Filling Title Block Information* (on page 267) for additional information.

9. Save the custom template as a *drawing* (see "Saving and Using a Custom Template as a Drawing" on page 266) or as a symbol (see "Saving and Using a Custom Template as a Symbol" on page 267).

12.5.2 Customizing an Existing Template

➢ ***To customize an existing template:***

1. Open a new drawing workspace. The **New Sheet Properties** dialog appears.
2. Select the **Template** option and select a template from the list. To open a custom template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog.

   Note: If you have previously **browsed** to another template folder, you can use the **Default** button to reset the template list back to the system template folder.

3. Click **OK** in the Sheet Properties dialog.
4. If the **Fill In Text** dialog appears, complete any applicable fields.
5. Click **OK** when finished with the Fill In Text dialog. The **Insert Design** dialog appears.
6. Click **Cancel**.
7. Right-click the sheet in the Drawing Explorer and select **Activate Sketch on Sheet** from the pop-up menu.
8. Move the mouse pointer over the drawing border. The border is highlighted.
9. Right-click and select **Explode Symbol** from the pop-up menu. The template is exploded into individual segments.
10. Make the necessary changes to the template.
11. From the **Sketch** menu, select **Activate Sketch** to leave the sketch mode.
12. Save the custom template as a *drawing* (see "Saving and Using a Custom Template as a Drawing" on page 266) or as a *symbol* (see "Saving and Using a Custom Template as a Symbol" on page 267).

Note: When you use the template, all of the entities will be placed on the continuous layer. You can right-click on an entity and choose Explode Symbol to explode the format and all entities will revert to their original layer. Most of the time you will not need to do this. It is only necessary if you need to see the different line styles for the entities in the template.

12.5.3 Saving and Using a Custom Template as a Drawing

➢ ***To save a custom template as a drawing:***

1. From the **File** menu, select **Save As**. The Save As dialog appears.
2. Navigate to the desired location in the file system.
3. Specify a name for the drawing template.
4. Click **Save**.

➢ ***To use a custom template drawing:***

1. Start a new drawing.
2. In the **New Sheet Properties** dialog, choose **Template**, and select the **Browse** button.
3. In the **Custom Drawing Template** dialog, browse through your file system to find the desired template, which you have already saved, and click it.
4. Click **OK** in the **Custom Drawing Template** dialog (The OK button is slow to activate here).
5. Click **OK.**

6. You will be prompted to fill in any default and user added text if you put any text fields in your template. Enter the desired information; then click **OK**.

   Your drawing format will appear, with the prompt to select a model.

Note: The next time the **New Sheet Properties** dialog is invoked, the **Template** list will be populated with all the custom templates located in the folder you last browsed to in step 2 above. You can use the **Default** button to reset this list to the default template folder.

12.5.4 Saving and Using a Custom Template as a Symbol

➢ ***To save a custom template as a symbol:***

1. Select the **Activate Sketch** tool from the Sketching toolbar.
2. From the **Sketch** menu select **Create Custom Symbol**. The Create Custom Symbol dialog appears.
3. In the workspace, drag a selection box around the figures, fields, and labels that you want to include in the template. These elements appear in the **Figures to include** list.
4. In the dialog, move your cursor to **Anchor Point** and click in the **X** or **Y** boxes.
5. Click in the work area where you want the bottom left corner of the template to appear in a new Drawing workspace. The coordinates appear in the **Anchor Point** area and are relative to the origin. Maintaining a (0,0) anchor point is acceptable.
6. Click **OK**. The **Save Custom Symbol** dialog appears.
7. Browse to the appropriate save location and select a folder**.**
8. Specify a name for the drawing template
9. Click **Save**.

➢ ***To use a custom template symbol:***

1. Open a new drawing workspace. The **New Sheet Properties** dialog appears.
2. Select **Blank Sheet**, and the corresponding sheet size.
3. Click the **Create An Empty Drawing** checkbox.
4. Specify the drawing **Scale**.
5. Click **OK**. A blank drawing workspace appears.
6. From the **Sketch** menu select **Activate Sketch**.
7. From the **Sketch** menu select **Insert Custom Symbol**. The **Select Custom Symbol** dialog appears.
8. Browse appropriate location and select the custom symbol you created to use as a template.
9. Click **OK**.
10. In the work area, select a location corresponding to the anchor point defined in the custom template. If your anchor point was defined at (0, 0), pick the sketch node representing the origin of the sheet. The custom template appears in the work area.
11. If any fields were created with the custom template, a dialog appears containing the field properties. Enter the text associated with the fields.
12. Click **OK**.

12.5.5 Automatically Filling Title Block Information

In Alibre Design there are two types of fields: user input fields and Design Property fields. User input fields prompt the user during drawing creation to enter in data. This data then gets positioned and displayed according to the field location and settings. Design Property fields pull information directly from the part or assembly files and do not allow or require user intervention. This provides a high degree of consistency and accuracy since it reduces human error.

Any data located in the **File > Properties > General** tab of a part or assembly workspace can be used to automatically populate data in a drawing.

There are two ways you can use Design Property fields. The first is to create a custom template that contains them. The second is to create custom symbols that contain them, and insert the custom symbols into the drawing where needed.

➢ ***To create a Design Property field:***

1. Open the template you want to contain Design Property fields.

2. Determine the information you want to pull from the part or assembly files. Review what is available in the **File > Properties > General** tab of the part and assembly workspaces.

3. Right click on the white space of your template and select **Activate sketch on sheet**. You are now in Sketch Mode.

4. From the Sketch menu, select **Text > Field**. The Field Text dialog appears.

5. Select **Design Property Value** under the **Set Type** option.

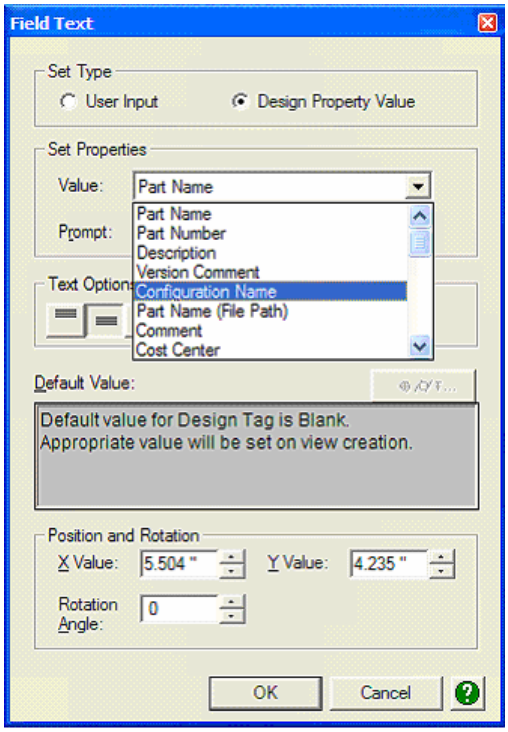


Figure 96: Field Text Dialog

6. Select the Design Property you want to use under the **Set Properties** section. In this example, we are using the Configuration Name.

7. Select the text alignment, font, and color options you would like. A preview of the output is shown as you change the options.

8. Position the field on the sheet by dragging it to the desired location or by typing in an exact value in the Position and Rotation section.

9. Click **OK**.

10. Save the drawing template.

11. When creating a new drawing, use the drawing template you just made as the template for the drawing. If you have filled out the data referenced by the fields in the **File > Properties > General** tab section of the part and assembly files you are projecting, that data will automatically be filled in upon view creation.

Note: All the fields on a sheet can reference a maximum of one design, and they all must reference the same design. If, for example, you have inserted a view of a part and of an assembly on the same sheet, you must pick which design will populate all the Design Property fields on that sheet. This is discussed in the next section.

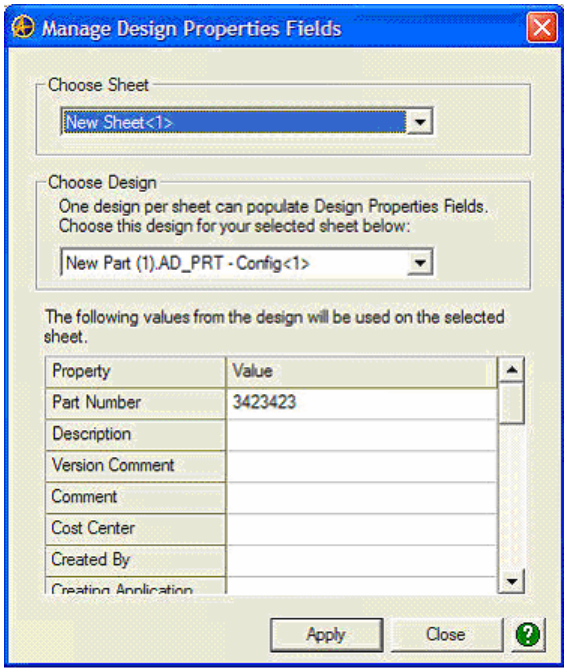
12.5.6 Managing Design Property Value Fields

If your template uses Design Property Value fields, or if you have inserted Custom Symbols containing Design Property Value fields, you may need to manage them. Managing means selecting which design on any particular sheet will be used to fill out Design Property Fields on that sheet.

Note: Only one design per sheet may be used to populate Design Property Value fields.

➢ ***To select the design to populate Design Property Value fields***

1. Select the Manage Design Properties icon under the Reproject icon. The Manage Design Properties dialog displays.



2. Select the **Sheet** and Design combination you would like to use. You can also select **None** under **Choose Design** to ignore any Design Property Values on the selected sheet.

3. Review the available Design Properties of the design. This section is read-only and provides a sanity check of the values before using them.

4. Click **Apply**.

12.6 Annotations

You can insert various annotation types into a part, assembly, or drawing workspace to describe and clarify design and manufacturing information. You can insert notes, datums and datum targets, feature control frames, surface finishes, weld symbols, and balloon callouts.

To insert annotations in any workspace, from the **Insert** menu select **Annotations**, and then select from the available annotation types.

In a drawing workspace, the annotation tools are also displayed on the Detailing toolbar.

Note	insert a note annotation
Datum	insert a datum annotation
Datum Target	insert a datum target annotation
Feature Control Frame	insert a feature control frame notation
Surface Finish	insert a surface finish annotation
Weld	insert a weld symbol annotation
Callout	insert a balloon callout annotation

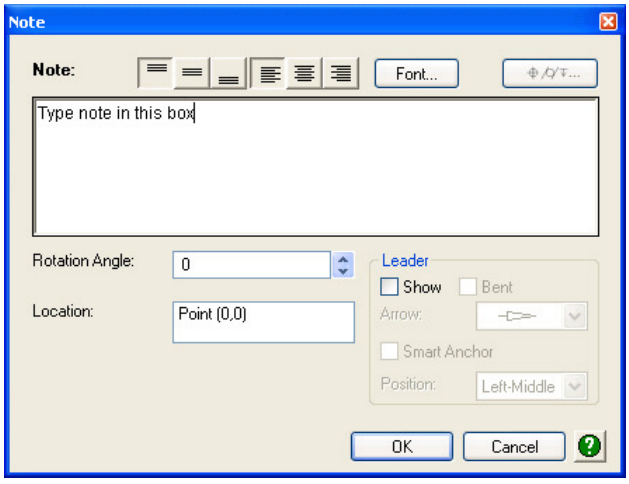
You can pre-define certain display characteristics for annotations, including arrow types and sizes, text font, and the shape of balloon callouts, in the **Annotations** tab of the **Drawing Properties** dialog.

12.6.1 Note

You can create a free-floating note or a note with a leader pointing to an edge or face. The note can contain both text and symbols.

➢ ***To insert a note:***

1. In any workspace, from the **Insert** menu select **Annotation > Note**; or in a drawing workspace, select the **Note** tool from the Detailing toolbar. The **Note** dialog appears.



2. In the **Note** area, type the annotation text. You may align the text vertically and horizontally by clicking the text alignment icons above the **Note** field.

3. Click **Font** to specify the font, font style, size, color, and effects.

4. To insert a symbol, click the **Symbols** button. The **Insert Alibre Design Symbols** dialog appears. Click a symbol to insert it. Click **Close** to close the symbols box.

5. Specify a **Rotation Angle** if required.

6. If you want to include a leader, click the **Show** option in the **Leader** area.

7. Select the **Bent** option if you want the leader line to have a short horizontal segment near the annotation.

8. From the **Arrow** pull-down menu, select the arrow type you want to use.

9. From the **Position** pull-down menu, select the position in which the text will be placed in relation to the leader.

10. Select **Smart Anchor** if desired. The smart anchor option will automatically adjust the position of the text if the leader is re-positioned. The smart anchor option overrides the position selection in step 9.

11. Click in the **Location** box and then click in the work area to select the location of the note. If you are using a leader, click a figure or face to attach the note to. The selection coordinates or displayed in the **Location** box.

12. Click **OK** to finish placing the note.

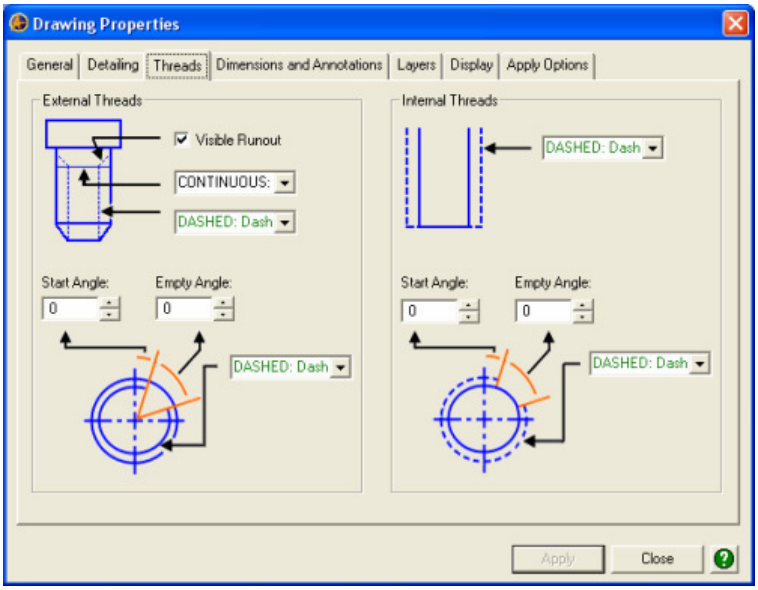
Note: In drawings, if you insert a note while a view is active, the note will be attached to that view. If you insert a note while the sheet is active, the note will be attached to the sheet.

To copy the annotation: click on it to select it (press and hold the Shift key to select multiple annotations). Then, either right click and choose **Copy**; or, from the **Edit** menu, select **Cop**y.

To paste the annotation: you can choose **Paste** from the **Edit** menu, or you can move your mouse to the location you want to paste the annotation and press **CTRL + V** on your keyboard.

12.6.2 Threads and Callouts

Internal and external thread information, if applied in the 3D design, will automatically be called out upon creation of views in the 2D drawing. Cosmetic threads, also displayed by default, are represented graphically by dashed lines.



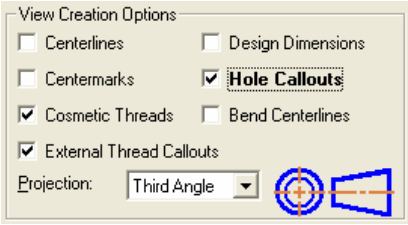
The default callout for holes includes the number of identical holes, the type of thread, and the thread depth.

Setting hole callout and thread default options

You can control whether hole callouts and internal and external cosmetic threads are automatically created with new views.

➢ ***To set default view creation options:***

1. From the **File** menu, select **Properties**.
2. On the **Detailing** tab, in the View Creation Options section, check on all of the options that you want to include by default in all newly created views.



Setting hole callout and thread options during view creation

You can also choose to include the hole callouts and cosmetic threads during the creation of the view in the Standard Views Creation dialog.

1. From the **Insert** menu, select **Standard Views**.
2. Fill out the information in the Standard Views Creation dialog as necessary.
3. Click the **More Options** button to see the View Creation options.
4. Check the Cosmetic Thread and Callout options as desired to include them automatically.
5. Click **OK** to continue with the view creation.

Manually adding, editing, or deleting hole callouts, cosmetic threads, and external thread callouts

➢ ***To manually apply hole or external thread callouts:***

1. Move the cursor over the hole or external thread. The item is highlighted.
2. Right-click the item and select **Insert Hole Callout** or **Insert External Thread Callout** from the pop-up menu. The callout is displayed.

   OR

3. Select the item.
4. From the **Insert** menu, select **Hole Callout** or **External Thread Callout**.

Note: You can show the callouts for an entire view by right-clicking a view in the Drawing Explorer, or right-click a view in the work area and selecting the appropriate option.

➢ ***To edit the hole callout:***

1. Move the cursor over the hole callout. The cursor changes and displays the annotation symbol.
2. Right-click the hole callout and select **Edit** from the pop up menu. The **Hole Callout** dialog appears.
3. Modify the **Callout Note** as necessary.
4. Modify the **Leader** parameters as necessary.
5. Click **OK** when finished.

Note: You must edit external thread callouts from the Part workspace.

➢ ***To manually apply cosmetic threads:***

1. Move the cursor over hole or over an edge of the cylinder that has threads.
2. Right-click and select **Insert Cosmetic Threads** from the pop-up menu. The cosmetic threads are displayed for the selected hole or thread cylinder.

   OR

3. Select the hole or cylinder.
4. From the **Insert** menu, select **Cosmetic Thread for View**.

Note: You can show the cosmetic threads for an entire view by right-clicking a view in the Drawing Explorer, or right-click a view in the work area and select **Insert Cosmetic Threads**.

➢ ***To delete a cosmetic thread:***

1. Move the cursor over the cosmetic thread symbol. The cursor changes and displays the annotation symbol.
2. Right-click the cosmetic thread and select **Delete** from the pop up menu.

Changing the appearance of cosmetic threads

You can change the appearance of internal and external cosmetic threads.

➢ ***To change the global thread appearance:***

1. Select the **File > Properties > Threads** tab.

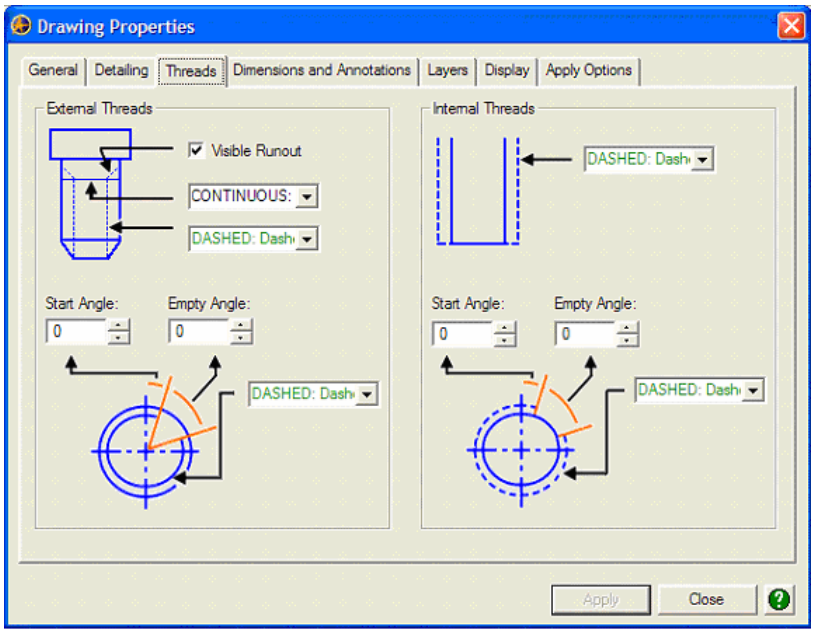


Figure 97: Drawing Properties Dialog-Threads Tab

2. Set the layer styles you want for the components of the internal and external threads.
3. Set the size of the empty angle you want, as well as the starting location for the empty portion of the head on view, by setting the **Start Angle** and Empty Angle.
4. Set whether or not you want external thread run-outs to be shown.

➢ ***To change individual thread appearance:***

1. Right click on any cosmetic thread.

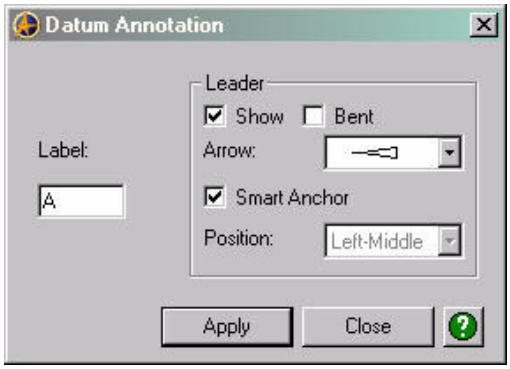
2. Select **Edit**. A dialog displays allowing you to override the global style shown above. The dialog will be identical to the internal thread or external thread portion of the dialog pictures shown previously, depending on the type of thread you are editing.

12.6.3 Datums

In part or assembly workspaces, you can attach datum annotations to faces of models (but not to vertices or edges). In drawing workspaces, you can place datum annotations at any location.

➢ ***To insert a datum annotation:***

1. In any type of workspace, from the **Insert** menu select **Annotation** > **Datum**. In a drawing workspace, select the **Datum** tool from the Detailing toolbar. The **Datum Annotation** dialog appears.



2. In the **Datum Label** box, specify the letter with which you want to start the datum series.

3. Select the **Show** option if want to use a leader with the datum. Select the **Bent** option if you want the leader line to have a short horizontal segment near the annotation.

4. From the **Position** pull-down menu, select the position in which the text will be placed in relation to the leader.

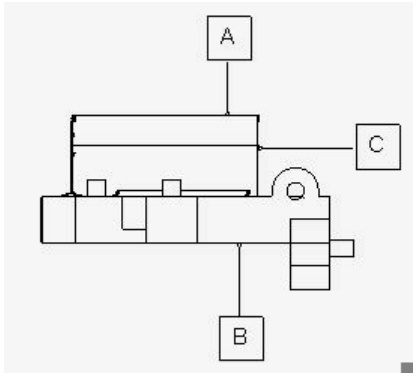
5. Select **Smart Anchor** if desired. The smart anchor option will automatically adjust the position of the text if the leader is re-positioned. The smart anchor option overrides the position selection in step 4.

6. From the **Arrow** pull-down menu, select the type of arrow to use with the leader.

7. In the work area, select the face, edge, or view to attach the annotation. If a leader is used, two clicks are required.

8. Continue to click to populate a series of datums.

9. Click **Apply** and **Close** when finished.



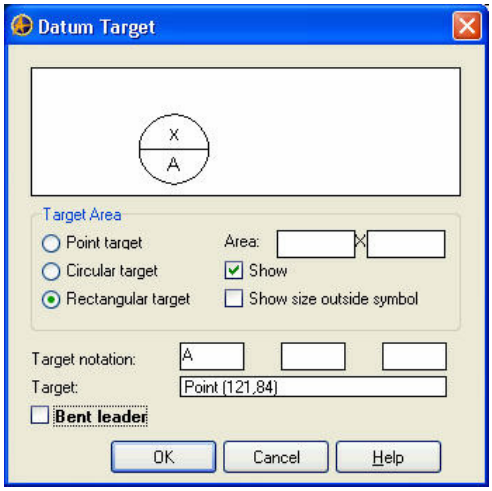
Datums can be resized and repositioned by clicking the text and dragging. This will change the length of the leader, as well as reposition the datum box. Datums can also be moved to another location by clicking the arrow, as opposed to the text.

12.6.4 Datum Targets

In part or assembly workspaces, you can attach datum target annotations to faces of models (but not to vertices or edges). In drawing workspaces, you can place datum annotations at any location.

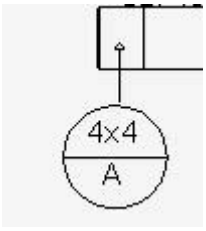
➢ ***To insert a datum target:***

1. In any type of workspace, from the **Insert** menu select **Annotation** > **Datum Target**. In a drawing workspace, select the **Datum Target** tool from the Detailing toolbar. The **Datum Target Annotation** dialog appears.



   The top area of the Datum Target dialog previews the annotation as you build it.

2. Select the target type: **Point**, **Circular**, or **Rectangular**.
3. In the **Area** fields, specify the target size.
4. Select the **Show** option to show or hide the datum target.
5. Select the **Show size outside symbol** option if desired.
6. In the **Target notation** fields, specify the datum reference label(s).
7. Click in the **Target** field.
8. In the work area, select the face, edge, or view to attach the annotation. Drag the target away from the selection to lengthen the leader.
9. Select the **Bent Leader** option if necessary.
10. Click to anchor the annotation.
11. Click **OK**.

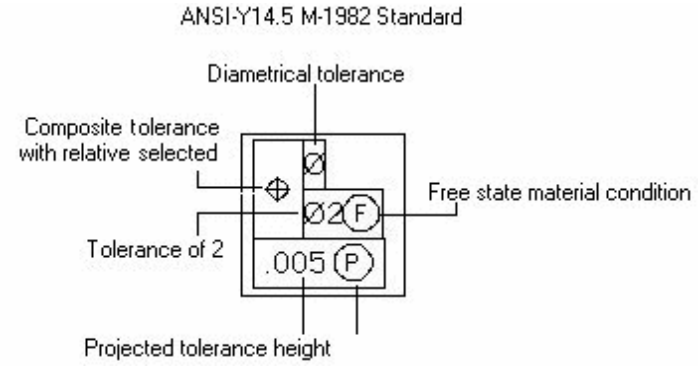


The datum target symbol can be repositioned by clicking the text and dragging. This will change the length of the leader, as well as reposition the datum target symbol. Datum targets can also be moved to another location by clicking the arrow, as opposed to the text.

12.6.5 Feature Control Frames

The geometric tolerance annotations let you specify a reference frame that contains all the geometric tolerance information for a selected surface or feature. The annotations support both the ANSI Y14.5 M-1982 and the 1994 standards.

Example



You also include datum references if the geometric tolerance is related to a datum.

You reference primary, secondary, and tertiary datum reference frames with the following material conditions for each geometric tolerance:

(M) MMC-maximum material condition

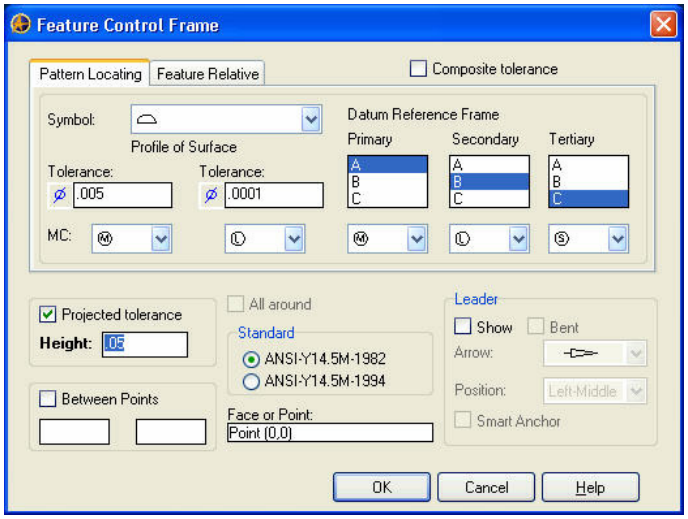
(L) LMC-least material condition

(S) RFS-regardless of feature size

(F) Free State-not limited by state position.

➢ ***To create a feature control frame:***

1. In any type of workspace, from the **Insert** menu select **Annotation** > **Feature Control Frame**. In a drawing workspace, select the **Feature Control Frame** tool from the Detailing toolbar. The **Feature Control Frame** dialog appears.



2. If the geometric tolerance is related to a datum, specify as many as three datum references that form the **Datum Reference Frame**. For the primary, secondary, and tertiary datums:
    - Click the **Datum** reference letter. (The reference letters were created when you inserted the Datum annotations.)
    - Click the **MC** arrow for the datum, and select the material condition for that datum.
3. Click the **Symbol** arrow, and select one or more of the displayed tolerance symbols.

    The geometric tolerance symbols indicate controls for form, profile, orientation, location, and runout.

Symbol	Control	Type
∠	Orientation	Angularity
○	Form	Circle
◎	Location	Concentricity
⌭	Form	Cylindricity
⏥	Form	Flatness
//	Orientation	Parallelism
⊥	Orientation	Perpendicularity
⌖	Location	Position
⌒	Profile	Line Edge
⌓	Profile	Surface
↗	Runout	Simple
—	Form	Straightness
⌯	Locations	Symmetry
⌰	Runout	Total

4. Specify the allowed **Tolerance** values.
5. Select the **Diametrical Tolerance** symbol if the tolerance is associated with a diameter zone.
6. Select a material condition from the **MC** pull-down menu for each tolerance value.
7. Repeat steps 2-6 on the Feature **Relative** tab if you want to specify a stacked feature control frame. This allows you to see a different symbol for each row. If you want a composite tolerance instead (one symbol shared across rows), check the **Composite Tolerance** checkbox.
8. Select the **Projected Tolerance** option if required and specify the **Height** of the projected tolerance zone.
9. Select the **Standard** that you want to use for the tolerance symbol. Alibre Design supports ANSI Y14.5 M-1982 and 1994. The preview shows the symbol for the selected standard.
10. Select the **Between Points** option to call out a tolerance between two points. Enter labels for the two points.
11. Select the **Show** option to include a leader, the **Bent** option if desired, and select an **Arrow** type.
12. Select **Smart Anchor** if necessary.
13. Move the mouse in the work area. A preview of the annotation appears attached to the mouse pointer. Left click once to place the leader line. Left click again to place the annotation. The text appears in light blue.
14. Click **Apply** to accept the annotation placement (or you can double-click). The dialog remains open so you can continue to place additional annotations.

Before choosing Apply, you can reposition the annotation while it is light blue in color. To do this, left click on the leader line or the text, whichever you want to move, and release the mouse button. Move the mouse to reposition. Left click again to place it.

15. Click **Close** to exit the dialog.

Note: You can copy and paste feature control frame annotations on the same sheet or paste them on another sheet in the same drawing file.

To copy the annotation: click on it to select it (press and hold the Shift key to select multiple annotations). Then, either right click and choose **Copy**; or, from the Edit menu, select **Copy**.

To paste the annotation: you can choose **Paste** from the **Edit** menu, or you can move your mouse to the location you want to paste the annotation and press **CTRL + V** on your keyboard.

12.6.6 Surface Finish Symbol

In drawing, part, or assembly workspaces you can specify the surface texture of a face by using a **Surface Finish Symbol**. In a part or assembly, you can attach annotations to solids. In a drawing, you can attach annotations to any view, or to the projected edge of a model.

➢ ***To create a surface finish symbol:***

1. In any type of workspace, from the **Insert** menu select **Annotation** > **Surface Finish**. In a drawing workspace, select the **Surface Finish** tool from the Detailing toolbar. The **Surface Finish** dialog appears.
2. From the **Symbol** pull-down menu, select the machining method for the surface finish.
3. From the **Lay Direction** pull-down menu, select the direction of the surface pattern.
4. In the **Roughness** area:
    - Specify a value for the **Maximum** allowable height deviation from the surface mean plane.
    - Specify a value for the **Minimum** allowable height deviation.
    - Specify a value for the average **Spacing** of roughness peaks.
    - Specify a value for the roughness **Sampling** length.
5. Select the symbol **Standard** that you want to use. Alibre Design supports ANSI Y14.16, ISO 1302, and JIS Symbols.
6. In **Material Removal**, specify the value for the amount of stock to be removed by the machining method that you selected.
7. In the **Waviness** area:
    - Type a value for the **Waviness** (peak-to-valley height) of the waves.
    - Type a value for the **Spacing** between adjacent peaks.
8. If you want to specify the **Production Method** to be used for the surface finish, type it in the box provided.
9. If you want to include a leader:
    - Select **Show**.
    - Select **Bent** if you want the line to have a short horizontal segment near the annotation.
    - Click the **Arrow** down arrow, and select a style.
10. Move the mouse in the work area. A preview of the annotation appears attached to the mouse pointer. Left click once to place the leader line. Left click again to place the annotation. The text appears in light blue.
11. Use the **Rotation Angle** to change the angle at which the symbol is displayed.
12. Check the **Flip Text** box if you desire to flip the text 180°.

    Note: The Flip Text box will automatically become checked if the rotation angle goes above 90°. However, you can uncheck the box if you do not want the text flipped.

13. Click **Apply** to accept the annotation placement (or you can double-click). The dialog remains open so you can continue to place additional annotations.

Before choosing Apply, you can reposition the annotation while it is light blue in color. To do this, left click on the leader line or the text, whichever you want to move, and release the mouse button. Move the mouse to reposition. Left click again to place it.

14. Click **Close** to exit the dialog.

Note: You can copy and paste surface finish annotations on the same sheet or paste them on another sheet in the same drawing file.

To copy the annotation: click on it to select it (press and hold the Shift key to select multiple annotations). Then, either right click and choose **Copy**; or, from the **Edit** menu, select **Copy**.

To paste the annotation: you can choose **Paste** from the **Edit** menu, or you can move your mouse to the location you want to paste the annotation and press **CTRL + V** on your keyboard.

12.6.7 Weld Symbol

➢ ***To create a weld symbol:***

1. In any type of workspace, from the **Insert** menu select **Annotation** > **Weld**. In a drawing workspace, select the **Weld** tool from the Detailing toolbar. The **Weld** dialog appears.
2. Click the **Far** or **Near** tab, depending on where you want the annotation placed in relation to the design.
3. From the **Finishing method** pull-down menu, select the method that you want to specify.
4. From the **Contour** pull-down menu, select the shape that you want for the weld surface.
5. Specify a **Groove angle** value in degrees.
6. Specify a **Root opening** value.
7. In the **Weld symbol** area, select the weld symbol from the pull-down menu. You can also type text into the boxes on both sides of the weld symbol.
8. In the **Joint with spacer** area, select the spacer type from the pull-down menu.
9. Select the applicable weld placement options:
    - **All around**
    - **Field or site weld**
    - **Display pointing down** (enabled if you select Field or site weld)
    - **Stagger weld** (enabled if you select a fillet for both the Near and Far tabs)
10. In the **Specification process** area, type any additional instructions to be included with the weld symbol.
11. If you are specifying both a **Near** and a **Far** weld, click the other tab, and repeat the steps for the other weld.
12. If you want to include a leader:
    - Select **Show**.
    - Select **Bent** if you want the line to have a short horizontal segment near the annotation.
13. In **Arrow**, select an arrow style.
14. Move the mouse in the work area. A preview of the annotation appears attached to the mouse pointer. Left click once to place the leader line. Left click again to place the annotation. The text appears in light blue.
15. Click **Apply** to accept the annotation placement (or you can double-click). The dialog remains open so you can continue to place additional annotations.

Before choosing Apply, you can reposition the annotation while it is light blue in color. To do this, left click on the leader line or the text, whichever you want to move, and release the mouse button. Move the mouse to reposition. Left click again to place it.

16. Click **Close** to exit the dialog.

Note: You can copy and paste weld annotations on the same sheet or paste them on another sheet in the same drawing file.

To copy the annotation: click on it to select it (press and hold the Shift key to select multiple annotations). Then, either right click and choose **Copy**; or, from the **Edit** menu, select **Copy**.

To paste the annotation: you can choose **Paste** from the **Edit** menu, or you can move your mouse to the location you want to paste the annotation and press **CTRL + V** on your keyboard.

12.6.8 Editing and Deleting Annotations

You can edit or delete an annotation anytime after it has been created and placed.

- ***To edit or delete an annotation:***

1. Select the **Select** tool from the View toolbar.

2. Move the cursor over the annotation. The annotation is highlighted and the cursor includes the symbol icon when it is over the annotation that can be edited or deleted.

3. Right-click the annotation and select **Edit** or **Delete** from the pop-up menu. In the **Edit** case, the **Annotation** dialog appears.

4. Make the necessary changes to the annotation properties.

CHAPTER 13

Bills of Material

You can create a bill of material (BOM) for an assembly, as well as a part, if necessary. You can create a custom BOM, or create a new BOM from a template.

The bill of material is fully associative to the assembly and/or drawing. A change made in the assembly (e.g. adding or removing parts and subassemblies) is automatically applied to the BOM.

You can launch the BOM workspace directly from the drawing. Changes made in the BOM workspace will be updated automatically in the drawing. Manual changes to the BOM are not reflected in the associated assembly (or part).

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Creating Bills of Material 282
Working With a BOM in a Drawing 283
Working in a BOM Workspace 290

13.1 Specifying BOM Data

You can specify BOM related properties for a part or for an assembly that you want to treat as a part for BOM purposes. Consequently, these properties can be displayed in a BOM of any assembly that contains the part or assembly.

- ***To save BOM data with a part:***

1. From the **File** menu, select **Properties**. The **Design Properties** dialog appears.
2. Select the **General** tab.
3. Scroll through the **Property** list to find the applicable BOM property.
4. To enter BOM property data, click in the corresponding value field. The cursor appears.
5. Type in the appropriate text.
6. Continue specifying value fields as required.
7. Click **Apply**.
8. Click **Close**.
9. Save the part.

When you add the part to an assembly, and subsequently create a BOM, the BOM data will automatically be displayed.

- ***To save BOM data with an assembly:***

Follow the procedure above for a part but also check the option **Treat as part in BOM**, which is found in the **General** tab of the **Design Properties** dialog.

Now, this assembly will be treated as a part whenever it is encountered in a BOM. It will appear as a single item in the BOM. Also, the BOM properties assigned to it will be reported in the BOM. The parts contained in this assembly will **not** appear as separate items in the BOM.

13.2 Creating Bills of Material

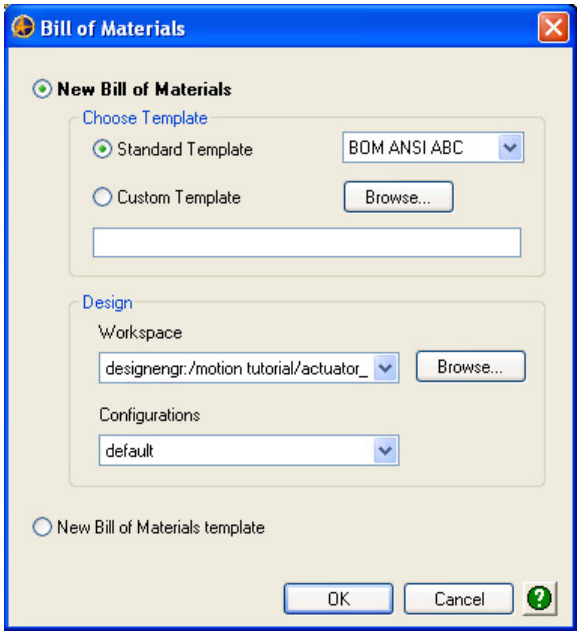
You can create a new Bill of Materials for an assembly, and a part, if necessary. You must first create the design and save it before a BOM can be created.

13.2.1 Creating a New BOM

➢ ***To create a new Bill of Materials:***

1. In the Home window or any workspace, from the **File** menu, select **New** > **Bill of Materials**; or, select the **Bill of Materials** icon from the main toolbar.

2. The Bill of Materials dialog appears.



3. Select the **New Bill of Materials** option.

4. In the **Choose Template** area, select the **Standard Template** or **Custom Template** option.

5. If you are using a Standard Template, select the appropriate template size as well.

Note: Alibre Design includes the BOM ANSI ABC and BOM ANSI D&E bill of materials templates ready for use. The BOM ANSI ABC template is for use with drawing templates ANSI A Portrait, ANSI A Landscape, ANSI B, and ANSI C. The BOM ANSI D&E template is for use with drawing templates ANSI D and ANSI E.

6. If you are using a Custom Template, click **Browse**. The **Select Bill of Materials Template** dialog appears. Select the appropriate template and click **OK**.

7. In the **Bill of Materials** dialog **Design** area, click **Browse**. The **Choose Design Part or Assembly** dialog appears.

8. Select the assembly or part for which you want a bill of materials.

9. Click **OK** in the Choose Design Part or Assembly dialog.

10. Click **OK** in the Bill of Materials dialog.

11. The BOM workspace appears containing the BOM data. When using a new BOM template, the first row will be blank by default. The default headers are **Item Number**, **Part Number**, **Quantity**, and **Part Name**.

12. *Modify the BOM* (see "Working in a BOM Workspace" on page 290) as required.

13. From the **File** menu, select **Save**; or select the **Save** tool from the main toolbar. The **Save** dialog appears.

14. In the Document Browser, select the location in which you want to save the BOM.

15. Enter the BOM **Name**.

16. Click **Save**. The BOM can now be opened independently or *inserted into a drawing* (see "Inserting a BOM View into a Drawing" on page 283) if necessary.

13.2.2 Creating a Custom BOM Template

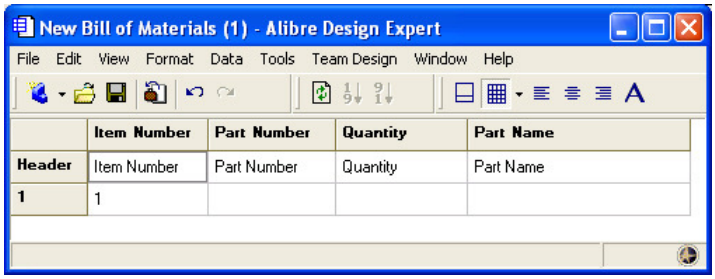
You can create a custom BOM template to meet your own design, purchasing, and production requirements and specifications.

➢ ***To create a custom BOM template:***

1. In the Home window or any workspace, from the **File** menu, select **New > Bill of Materials**; or select the **New Bill of Materials** icon from the main toolbar. The Bill of Materials dialog appears.

2. Select the **New Bill of Materials template** radio button**.**

3. Click **OK**. A New Bill of Materials workspace appears. The workspace contains one empty row by default. The default column headers are **Item Number**, **Part Number**, **Quantity**, and **Part Name**.



Note: If you choose to leave a blank row in the custom template, the blank row will be listed first any time you use the custom template. Delete the blank row if you do not want to include it in the custom template.

4. *Modify the table* (see "Working in a BOM Workspace" on page 290) as necessary to meet your requirements.

5. Select the **Save** tool from the Standard toolbar; or from the **File** menu, select **Save**. The **Save** dialog appears.

6. In the Document Browser, select the location in which you want to save the custom template.

7. Specify a **Name** for the custom template.

8. Select **Alibre** as the **Save as** type.

9. Click **Save**.

13.3 Working With a BOM in a Drawing

13.3.1 Inserting a BOM View into a Drawing

You can insert a new BOM view or existing BOM view into a drawing.

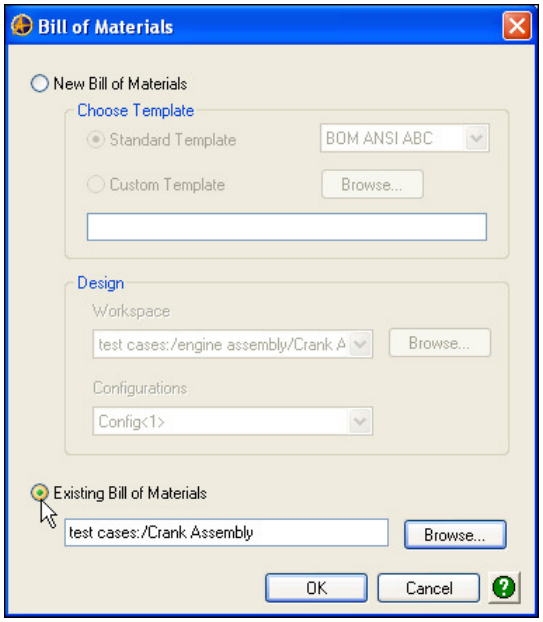
Note: You can only insert one BOM view per drawing. However, you can insert the BOM view into any drawing sheet.

When you insert a BOM view into a drawing, you are automatically linking the BOM to the drawing. Linking a BOM to a drawing creates an association between the BOM data and the drawing itself. You can *link a BOM to a drawing* (see "Linking a BOM to a Drawing" on page 285) without actually inserting the view into a sheet.

➢ ***To insert an existing BOM view into a drawing:***

1. Select the **Insert Bill of Materials** tool from the Detailing toolbar; or from the **Insert** menu, select **Bill of Materials View**.

   The **Bill of Materials** dialog appears.

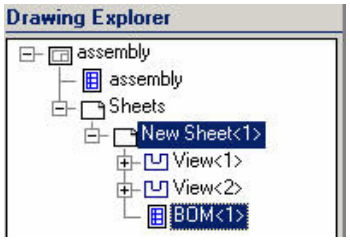


2. Click the **Existing Bill of Materials** radio button.

3. Click **Browse**. The **Choose Design Part or Assembly** dialog appears. In the Document Browser, navigate to the location containing the BOM.

4. Select the BOM item and click OK.

5. Click **OK** in the Bill of Materials dialog. A preview of the BOM view appears in the work area and is listed in the Drawing Explorer with the other views..



6. Move the cursor to position the BOM view on the sheet and click to place the view.

7. You can move a BOM view just like any other drawing view.

➢ ***To insert a new BOM view into a drawing:***

1. Select the Insert Bill of Materials tool from the Detailing toolbar; or from the **Insert** menu, select **Bill of Materials View**. The Bill of Materials dialog appears.

2. Click the **New Bill of Materials** radio button.

3. Select a Standard Template.

   Or

   Select a Custom Template. Click **Browse** to select the custom template.

4. In the Design area, click **Browse**. The Choose Design Part or Assembly dialog appears.

5. In the Document Browser, navigate to the location containing the design.

6. Select the design and click **OK**.

7. Click **OK** in the Bill of Materials dialog. A preview of the BOM view appears in the work area and is listed in the Drawing Explorer in the sheet view list.

8. Move the cursor to position the BOM view on the sheet and click to place the view.

13.3.2 Linking a BOM to a Drawing

Linking a BOM to a drawing creates an association between the BOM data and the drawing itself. You can link a BOM to a drawing without actually inserting the BOM view into a sheet. This is useful if you want to display item callouts in a drawing but do not want to display the BOM data in the sheet.

Before you can insert callout balloons, you must either link a BOM to a drawing or insert a BOM view into the drawing. Inserting a BOM view into a drawing automatically links the BOM to the drawing.

- ***To link a BOM to a drawing:***

1. From the **Tools** menu, select **Bill of Materials** > **Link**; or in the Drawing Explorer, right-click the drawing name and select **Link Bill of Materials** from the pop-up menu. The **Bill of Materials** dialog appears.

2. To link a new BOM, select the **New Bill of Materials** option. Specify the **Standard Template** or **Custom Template** option, and select a size or template accordingly. Click **Browse** to select the applicable design.

3. To link an existing BOM, select the **Existing Bill of Materials** option.

4. Click **Browse**. The **Choose Design Part or Assembly** dialog appears.

5. In the Document Browser, navigate to the location of the BOM.

6. Select the applicable BOM in the item list.

7. Click **OK**. The BOM name appears in the Bill of Materials dialog.

8. Click **OK**. The BOM item appears in the Drawing Explorer under the drawing name.

You can insert callout balloons after the BOM has been linked to the drawing.

13.3.3 Unlinking a BOM from a Drawing

Unlinking a BOM from a drawing removes all association between the BOM and drawing. If the BOM view has been inserted into the sheet, unlinking a BOM will delete the BOM view from the sheet automatically.

- ***To unlink a BOM from a drawing:***

1. In the Drawing Explorer, right-click the BOM item beneath the drawing name and select **Unlink** from the pop-up menu; or from the **Tools** menu, select **Bill of Materials** > **Unlink**. If a BOM view exists in the drawing, the **Unlinking Bill of Materials** dialog appears.

2. In the **Unlinking Bill of Materials** dialog, click **Yes**. If applicable, the BOM view is deleted from the drawing sheet, and the association between the BOM and the drawing is broken.

13.3.4 Editing a BOM

You can open the BOM workspace directly from the drawing and subsequently edit the BOM attributes.

➢ ***To edit the BOM from the drawing:***

1. Select the **Select** tool from the View toolbar.

2. Move the cursor over the BOM view in the work area and double-click; or in the Drawing Explorer, right-click the BOM item and select **Edit Bill of Materials** from the pop-up menu; or in the Drawing Explorer, double-click the BOM item. The BOM workspace appears.

3. In the BOM workspace, edit the BOM as necessary. Refer to *Assembly Constraints* (on page 213) for information related to working in a BOM workspace. The BOM view in the drawing will update automatically.

4. Close the BOM workspace when finished. You do not need to save the changes before you close the BOM workspace. Any changes made to the BOM will be saved when you save the drawing.

13.3.5 Moving the BOM View on the Sheet

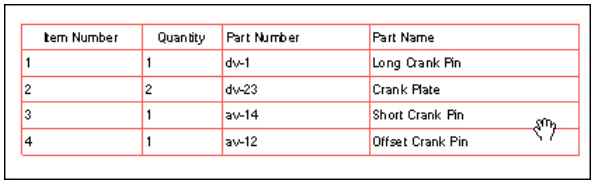
You can move a BOM view after it has been inserted into a sheet.

➢ ***To move a BOM view on a sheet:***

1. From the **Tools** menu, select **Selection Filters > Views**, if it is not already selected.

2. Select the **Select** tool from the View toolbar.

3. Move the cursor over the view in the work area. The view is highlighted and the cursor changes.



Item Number	Quantity	Part Number	Part Name
1	1	dv-1	Long Crank Pin
2	2	dv-23	Crank Plate
3	1	av-14	Short Crank Pin
4	1	av-12	Offset Crank Pin

4. Click and drag the view to the desired location on the sheet.

5. Release the mouse button to place the view.

13.3.6 Hiding the BOM View

You can hide the BOM view in a sheet.

➢ ***To hide a BOM view:***

1. In the Drawing Explorer, right-click the BOM item listed under the sheet and select **Hide** from the pop-up menu.

   Or

   Click the **Select** tool from the View toolbar.

2. Move the cursor over the BOM view in the work area, right-click, and select **Hide** from the pop-up menu.

The view is hidden in the work area and the associated text is dimmed in the Drawing Explorer.

➢ ***To show the BOM view:***

Right-click the dimmed BOM item in the Drawing Explorer and deselect **Hide** from the pop-up menu.

13.3.7 Deleting the BOM View

You can delete a BOM view from a sheet at anytime.

➢ ***To delete a BOM view:***

1. In the Drawing Explorer, right-click the BOM item listed under the sheet and select **Delete** from the pop-up menu.

   Or

   Click the **Select** tool from the View toolbar.

2. Move the cursor over the table in the work area, right-click, and select **Delete** from the pop-up menu.

The table is deleted from the work area and Drawing Explorer.

Note: The BOM is still associated with a drawing after you delete a table from a sheet. You must unlink the BOM from the drawing to remove all association between the BOM and drawing. Refer to *Inserting an Existing Design Into an Open Assembly* (on page 206) for information related to unlinking a BOM from a drawing.

13.3.8 Moving a BOM View to Another Sheet

You can move the BOM view from one sheet to another sheet in the drawing, if necessary.

➢ ***To move the BOM view from one sheet to another:***

1. Select the **Select** tool from the View toolbar.

2. Right-click the BOM view in the work area or the Drawing Explorer and select **Move** from the pop-up menu. The **Move** item is enabled only when the drawing contains at least two sheets. The **Select Target Sheet** dialog appears.

3. From the **Target Sheet** list, select the sheet to which you want to move the BOM view.

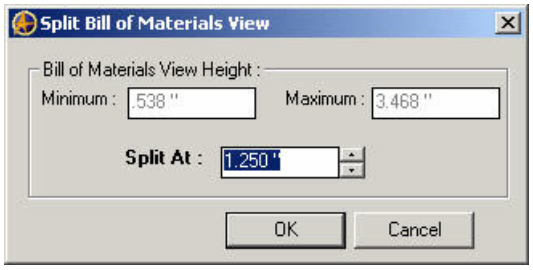
4. Click **OK**. The BOM view is listed under the target sheet in the Drawing Explorer and appears in the target sheet work area.

13.3.9 Splitting a BOM View

You can split a BOM view into multiple smaller views, if necessary. This is useful if a view is too long to fit onto a sheet.

➢ ***To split a view:***

1. From the **Tools** menu, select **Bill of Materials** > **Split View**; or right-click the BOM view in the work area or Drawing Explorer and select **Split View** from the pop-up menu. The Split Bill of Materials View dialog appears.



   The **Minimum** and **Maximum** values are specific to the table being split. The **Minimum** value represents the combined width of the header row and the widest row in the table. The **Maximum** value represents the combined width of the header row and all the rows in the table, except for the last row.

2. In the **Split At** field, specify a split value. This value must fall between the **Minimum** and **Maximum** view height values.

3. Click **OK**. The view is split into multiple views.

You can move the views independently on the sheet. You can also move individual views onto a different sheet, if necessary. However, if you delete or hide one view, the rest of the views will be deleted or hidden as well. If you add a row to the BOM after is has been split, the row will be added to the last BOM view.

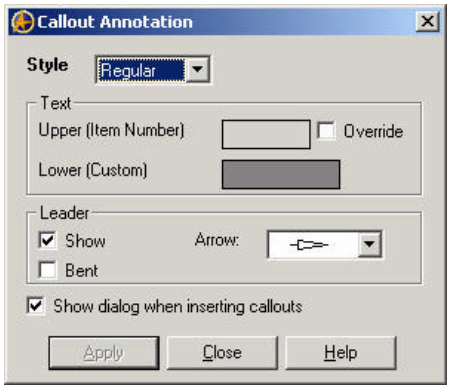
To restore the view back to its original configuration, enter a **Split At** value outside the **Minimum** - **Maximum** range.

13.3.10 Adding Callout Balloons

After you have linked a BOM with a drawing or inserted a BOM view into a drawing, you can add callout balloons to drawing views.

➢ ***To add callout balloons:***

1. Select the **Callout** tool from the Detailing toolbar; or from the **Insert** menu, select **Annotation** > **Callout**. The **Callout Annotation** dialog appears.



2. Select **Regular** or **Split** from the **Style** pull-down menu. The **Regular** style will, by default, display only the item number in the callout balloon. The **Split** style will divide the callout balloon into halves. The upper half displays the item number by default, and the lower half displays custom information.

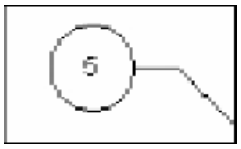


Figure 98: Regular Callout

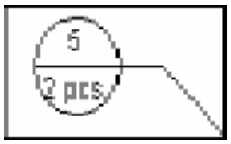


Figure 99: Split Callout

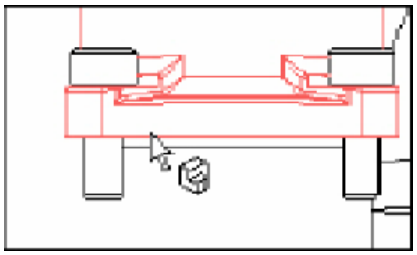
3. If necessary, select the **Override** option to manually enter an item number.

4. If the **Split** type was selected, enter custom text in **Lower (Custom)** text box area.

5. In the **Leader** area, select:

   - **Show** if you want to display a leader with the callout balloon.
   - **Bent** if you want to display a bent leader with the callout balloon.
   - An **Arrow** type from the pull down menu.

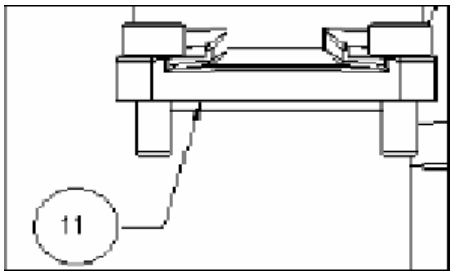
6. Move the cursor over a part in the drawing view. The part is highlighted.



7. Click once to create the callout balloon. The callout balloon appears.

8. Drag the balloon to the appropriate position and click to place.

9. Click **Apply** in the **Callout Annotation** dialog. The balloon is placed.



10. Continue to select parts and click **Apply** to add additional balloons.

11. Click **Close** when finished.

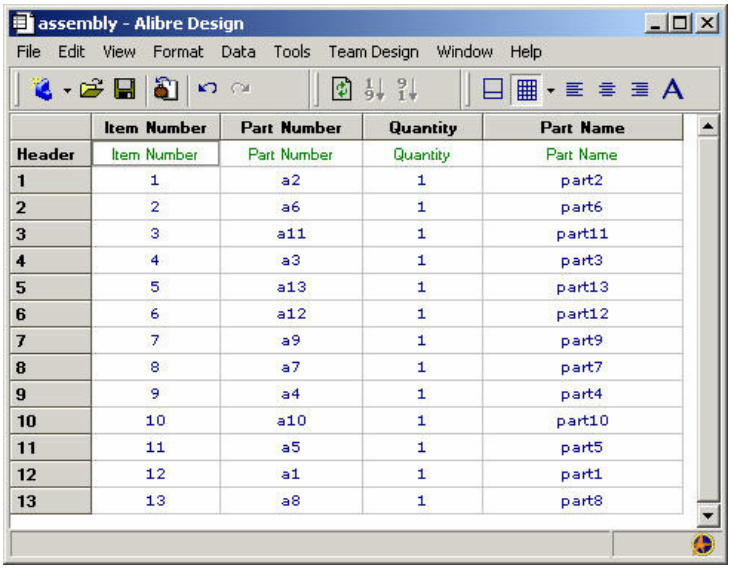
Notes:

The Callout Annotation dialog appears by default when you insert a callout. If desired, you can specify the callout settings you want to use in the dialog, and then deselect the **Show dialog when inserting callouts** option. You will then be able to insert callouts quickly without using the dialog. After selecting the **Callout** tool, simply click a part in a view to create the callout balloon and click to place it. To turn the dialog option back on, from the **Tools** menu, select **Options**. In the **General** tab, select the **Show dialog when inserting callouts** option.

You cannot copy and paste BOM callouts.

13.4 Working in a BOM Workspace

All work related to creating or editing a BOM is performed in a BOM workspace. The BOM workspace displays the bill of material data in tabular format similar to a spreadsheet.



	Item Number	Part Number	Quantity	Part Name
Header	Item Number	Part Number	Quantity	Part Name
1	1	a2	1	part2
2	2	a6	1	part6
3	3	a11	1	part11
4	4	a3	1	part3
5	5	a13	1	part13
6	6	a12	1	part12
7	7	a9	1	part9
8	8	a7	1	part7
9	9	a4	1	part4
10	10	a10	1	part10
11	11	a5	1	part5
12	12	a1	1	part1
13	13	a8	1	part8

In a BOM workspace you can:

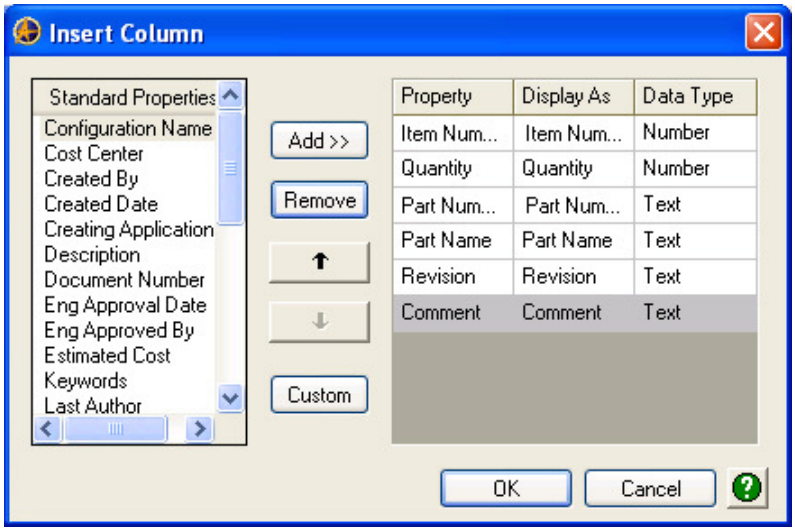
- Add/delete rows & columns
- Resize rows & columns
- Hide rows
- Change data and header font properties
- Automatically re-sequence data
- Override design values
- Set column header and data alignment
- Print BOM data
- Export BOM data to a .CSV file
- Append (add) rows
- Organize data by dragging and dropping columns and rows
- Change the table display orientation
- Sort data in ascending or descending order
- Control how a BOM will be displayed in a drawing

13.4.1 Adding and Deleting Columns in a BOM

You can add standard or custom columns to a BOM, as well as delete columns, as needed.

➢ ***To add a column to a BOM:***

1. From the **Edit** menu, select **Insert Column**; or right-click in the table area and select **Column** > **Insert** from the pop up menu. The **Insert Column** dialog appears.

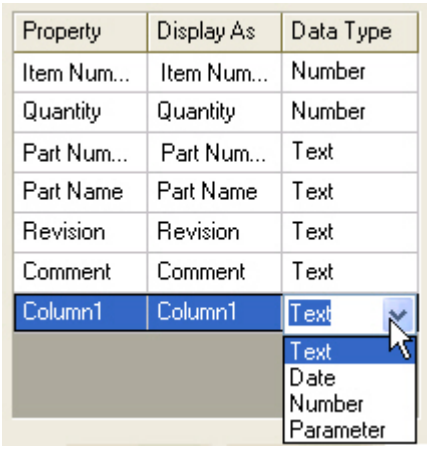


Note: In the Insert Column dialog, you can edit the fields under the Display As column. You cannot edit the **Property** or **Data Type** fields when using standard headers.

2. In the **Standard Properties** area, select a column header from the options listed.

3. Click the **Add** button to move the standard header to the column list.

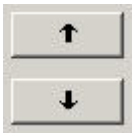
4. To create a column with a custom header, click the **Custom** button. The custom column is automatically added to the column list.



5. Edit the **Display As field** for the custom header as necessary.

6. In the **Data Type** column, select the column data format: **Text**, **Date**, **Number, or Parameter**.

7. Columns will be displayed in the BOM in the order in which they are listed in the Insert Column dialog. To move a column up or down in the list, select a row, and then click either the up arrow or down arrow.



8. Click **OK**.

➢ ***To delete a column from a BOM:***

1. Click the table header of the column you want to delete (indicated by the mouse pointer in the image below). The entire column is highlighted.

2. Right-click in the table area and select Column > Delete from the pop up menu; or from the **Edit** menu, select **Delete**.

13.4.2 Adding and Deleting a Row in a BOM

You can add rows in a BOM as needed. You can also delete a row at any time that has been manually inserted. However, in order to delete a row that was generated automatically from a design (i.e., a part in the assembly), you must first delete the associated part in the assembly. The Quantity value must then be updated to zero before the row can be deleted.

➢ ***To add a row:***

From the **Edit** menu, select **Append Row**; or right-click in the table area and select **Row > Append** from the pop up menu. A row is added to the end of the table.

➢ ***To delete a row:***

1. Click the table row number you want to delete. The entire row is highlighted.

2. Right-click in the table area and select **Row** > **Delete** from the pop up menu; or from the **Edit** menu, select **Delete**.

13.4.3 Hiding a Row

You can hide rows in a BOM. Hidden rows are not displayed in the BOM view in the drawing.

➢ ***To hide a row:***

1. Click the table row number you want to hide. The entire row is highlighted.

2. Right-click in the table area and select **Row** > **Hide** from the pop up menu; or from the **Format** menu, select **Row** > **Hide**. The row becomes hidden.

Note: A distinct line is displayed between the rows that border above and below the row that has been hidden. In the illustration below, row 2 has been hidden.

Header	Item Number	Quantity	Part Number	Part Name
1	1	1	dv-1	Long Crank Pin
3	3	1	av-14	Short Crank Pin
4	4	1	av-12	Offset Crank Pin

➢ ***To display hidden rows:***

From the **View** menu, select **Hidden Rows**. The hidden row is displayed and the corresponding table row number field is orange.

Header	Item Number	Quantity	Part Number	Part Name
1	1	1	dv-1	Long Crank Pin
2	2	2	dv-23	Crank Plate
3	3	1	av-14	Short Crank Pin
4	4	1	av-12	Offset Crank Pin

➢ *To unhide a row:*

1. If hidden rows are displayed, select the table row number of the hidden row. The entire row is highlighted.

   Or

   If hidden rows are not displayed, select table row number above the hidden row, and then hold the **Shift** key and select the table row number below the hidden row. Both selected rows become highlighted.

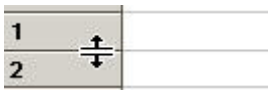
2. Right-click in the table area and select **Row** > **Unhide** from the pop up menu; or from the **Format** menu, select **Row** > **Unhide**.

13.4.4 Resizing Rows and Columns

You can resize rows and columns to customize the look of the table.

➢ ***To resize a row or column by dragging:***

1. In the row number or column header area, move the cursor near the edge of the row or column you want to resize. The cursor changes appearance.



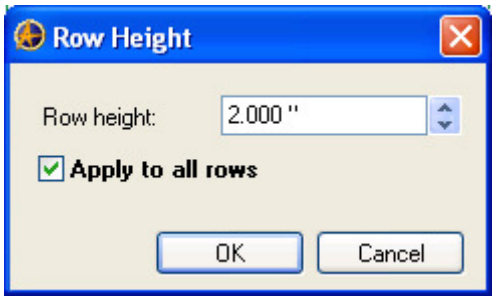
2. Click and drag the row or column border to change the respective width or height.

3. Release the mouse button when finished resizing.

➢ ***To resize a row by specifying a row height value***

1. Click the row number you want to resize. The entire row is highlighted.

2. Right-click in the table area and select **Row** > **Height** from the pop up menu; or from the **Format** menu, select **Row** > **Height**. The **Row Height** dialog appears.



3. Specify the **Row height** value.

4. Check the **Apply to all rows** option if desired.

5. Click **OK**.

➢ ***To automatically adjust a column's width:***

1. In the main column header area, move the cursor over the right edge of the column you want to automatically resize.
2. Double-click the right edge. The column width automatically adjusts to the widest field in the column.

   Or

   Click the table Header of the column you want to automatically resize. The entire column is highlighted.
3. Right-click in the table area and select **Column** > **AutoFit** from the pop up menu; or from the **Format** menu, select **AutoFit Column**. The column width automatically adjusts to the widest field in the column.

13.4.5 Adjusting Column Header and Data Alignment

You can adjust the alignment of column headers and column data independently or together.

➢ ***To adjust column header alignment:***

1. Select any field in the column in which you want to change the header alignment.
2. Right-click in the table area and select **Header Alignment** > **Left** or **Center** or **Right**.

   Or

   From the **Format** menu, select **Header Alignment** > **Left** or **Center** or **Right**.

   Or

   Select the **Align Left** , **Center** , or **Align Right** tool from the View toolbar.

   The column header alignment changes.

➢ ***To adjust column data alignment:***

1. Select any field in the column in which you want to change the data alignment.
2. Right-click in the table area and select **Data Alignment** > **Left** or **Center** or **Right**.

   Or

   From the **Format** menu, select **Data Alignment** > **Left** or **Center** or **Right**.

   Or

   Select the **Align Left** , **Center** , or **Align Right** tool from the View toolbar.

   The column data alignment changes.

➢ ***To adjust column header and data alignment together:***

1. Click the table header of the column you want align. The entire column is highlighted.
2. Right-click in the table area and select **Data Alignment** > **Left** or **Center** or **Right**.

   Or

   From the **Format** menu, select **Data Alignment** > **Left** or **Center** or **Right**.

   Or

   Select the **Align Left** , **Center** , or **Align Right** tool from the View toolbar.

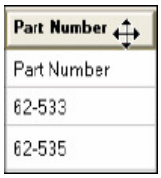
   The column header and data alignment changes.

13.4.6 Moving Rows and Columns in a Table

You can drag and drop rows and columns to reposition data within the table.

➢ ***To move a row or column:***

1. Move the cursor over a table column header or row number.



2. Click the column header cell or row number cell and drag to the new table position.

3. Release the mouse button to complete the move.

13.4.7 Sorting Data in Ascending or Descending Order

You can sort column data in ascending or descending order.

➢ ***To sort data in ascending or descending order:***

1. Click the table header of the column you want to sort. The entire column is highlighted.

2. To sort the column data in ascending order, select the **Sort Ascending** tool from the main toolbar; or from the **Data** menu, select **Sort** > **Ascending**.

   Or

   To sort the column data in descending order, select the **Sort Descending** tool from the main toolbar; or from the **Data** menu, select **Sort** > **Descending**.

   The data is sorted accordingly.

13.4.8 Changing the Header Display Orientation

You can change the table display so that the header is located at the bottom of the table and the row numbers increase going up the table. By default, the header is located at the top of the BOM table and the row numbers increase going down the table.

➢ ***To change the header display orientation:***

1. Select the **Bottom Up Display** tool from the View toolbar; or From the **View** menu, select **Bottom Up Display**. The column headers are positioned at the bottom of the table and the row numbers increase going up the table.

2. To return to the default orientation, repeat step 1.

13.4.9 Customizing Header and Data Font Properties

You can change the font properties associated with column headers and tabular data. You cannot change font properties for individual items in the table.

➢ ***To customize header font properties:***

1. Right-click in the table area and select **Header Font** from the pop-up menu; or from the **Format** menu, select **Header Font**. The **Font** dialog appears.

2. Modify the **Font**, **Font Style**, **Size**, **Effects**, **Color**, and **Script** as desired.

3. Click **OK**.

➢ ***To customize data font properties:***

1. Right-click in the table area and select Header Font from the pop-up menu; or from the **Format** menu, select **Header Font**. The Font dialog appears.

2. Modify the **Font**, **Font Style**, **Size**, **Effects**, **Color**, and **Script** as desired.

3. Click **OK**.

13.4.10 Overriding Design Values

When you create a BOM, the table contains information based on the design, e.g. part number, part name, quantity, etc. These items are referred to as **design values** since they are dictated by the design. You can manually override design values in a BOM workspace. Overriding a design value in the BOM workspace will not have an effect on the actual design.

➢ ***To override a design value:***

1. Select the field containing the design value that you want to override.

2. Change the value as necessary and press **Enter** on the keyboard. The cell containing the overridden value becomes blue.

10	11		a10	1	part10
11	12		a5	5	part5
12	13		a1	1	part1

➢ ***To restore a value to its design value:***

1. Select the cell containing the overridden value.

2. Right-click the cell and select **Use Design Valu**e from the pop-up menu; or from the **Edit** menu, select **Use Design Value**. The cell's value is restored to the value dictated by the design.

13.4.11 Modifying the BOM View Style

You can control how the BOM view is displayed in the drawing. You can choose to show or hide row and column lines, only column lines, only row lines, or no lines. The table style setting only applies to the BOM view in the drawing. Table lines are always visible in the BOM workspace regardless of which table style setting is used. By default, row and column lines are visible.

➢ ***To modify the table style:***

1. Click the **Options** arrow on the View toolbar to display the Table Style drop down toolbar.

2. From the toolbar, select:

   - **No Lines**: the BOM table will be displayed without lines.
   - **Row Lines**: the BOM table will be displayed with row lines only.
   - **Column Lines**: the BOM table will be displayed with column lines only.
   - **Row, Column Lines**: the BOM table will be displayed with column and row lines.

Or

1. From the **Format** menu, select **Table Style**.

2. Choose either **No Lines**, **Row Lines**, **Column Lines** or **Row**, **Column Lines**.

13.4.12 Resequencing Data

You can resequence (reorder) data in a table after you delete or move rows. Resequencing a BOM will reset the **Item Numbers** so that they are sequentially numbered correctly in the order listed.

➢ ***To resequence a BOM:***

From the **Data** menu, select **Resequence**. The **Item Numbers** are reordered.

	Item Number	Quantity	Part Number	Part Name
Header	Item Number	Quantity	Part Number	Part Number
1	1	1	a2	part2
2	7	1	a4	part4
3	2	1	a6	part6
4	9	1	a5	part5
5	4	1	a13	part13
6	5	1	a12	part12
7	8	1	a10	part10
8	10	1	a1	part1
9	6	1	a7	part7
10	11	1	a8	part8
11	12	1	a9	part9
12	13	1	a11	part11
13	3	1	a3	part3

Figure 100: BOM Table Before Resequencing

	Item Number	Quantity	Part Number	Part Name
Header	Item Number	Quantity	Part Number	Part Number
1	1	1	a2	part2
2	2	1	a4	part4
3	3	1	a6	part6
4	4	1	a5	part5
5	5	1	a13	part13
6	6	1	a12	part12
7	7	1	a10	part10
8	8	1	a1	part1
9	9	1	a7	part7
10	10	1	a8	part8
11	11	1	a9	part9
12	12	1	a11	part11
13	13	1	a3	part3

Figure 101: BOM Table After Resequencing

13.4.13 Updating the Table

You can have the assembly workspace open and the BOM workspace open simultaneously. Consequently, you can update a BOM table after making changes to an assembly. Before you can update the BOM, you first must save any changes made to the design. The design does not need to be open in order to update the table. You cannot update the BOM when it is being edited within the context of the drawing.

➢ ***To update the table:***

From the **Data** menu, select **Update Table**.

Or

Select the **Update Table** tool from the Edit toolbar.

The table is updated to reflect any recent changes made in the design.

13.4.14 Exporting a BOM

You can export a BOM table as a .csv file. You can open .csv files in any spreadsheet application or text editor.

➢ ***To export a BOM:***

1. Select the **Export File** tool from the Standard toolbar; or from the **File** menu, select **Export**. The **Export File** dialog appears.
2. Select the **Save in** location.
3. Specify a **File name**.
4. Select a **Save as type**.
5. Click **Save**.

13.4.15 Printing a BOM

You can print a BOM table by itself directly from the BOM workspace.

➢ ***To print a BOM table:***

1. From the **File** menu, select **Print**. The **Print** dialog appears.
2. Select the appropriate printer.
3. Specify the print layout.

CHAPTER 14

Opening and Saving

In Alibre Design, you can utilize the File System or the Alibre Vault (if you are licensed for it) to quickly open and save files.

In This Chapter

Opening Files 299
Saving Files 300

14.1 Opening Files

Alibre Design uses standard Windows dialogs for opening Alibre Design files. Depending on whether or not you are using the Vault, you may need to navigate to the Vault drive to find your files through this dialog. See *The Alibre Vault by M-Files* (on page 347) for details on how to enable the Vault within Alibre Design.

14.1.1 Opening Files from the File System

➢ ***To open any Alibre Design File:***

You can do any of the following:

- From any workspace, select **File > Open** and browse to find your file.
- Select the Open icon in any workspace.
- Select the Open icon on the Home Window

Note: You can also browse to your files through Windows Explorer and double-click them to open them in Alibre Design.

14.1.2 Opening Files from the Alibre Vault

You can access the Alibre Vault in several ways.

➢ ***To open any Alibre Design File from within Alibre Design:***

- From any workspace, select **File > Open** and browse to your Alibre Vault drive (usually M: by default).
- Select the Open icon in any workspace.
- Select the Open icon on the Home Window.

➢ ***Navigate to the Alibre Vault using Windows Explorer:***

1. Go to My Computer and select the M-Files drive (usually M:). Or, select the icon at the bottom right of the screen in the System Tray.
2. Select the Alibre Vault.
3. Browse to your files using the search filter or View filters.

14.2 Saving Files

The act of saving can have slightly different behavior depending on exactly what you are doing and the workspace in which you are doing it. Alibre Design files are "linked" to each other in most cases. For example, an assembly file is linked to each of the part files that are used in the assembly. A drawing of the assembly would be linked to the assembly file, as well as each of the part files used in the assembly.

14.2.1 General Saving Principles

Alibre Design will save the minimum number of items necessary to represent any changes you have made.

There are three basic types of saves:

1. **Save** - Saves the current design. If the item has never been saved, this will perform a Save As and ask you to name the file.

2. **Save As** - Creates a duplicate of the top level item you are working on. For example, if you are working on a part, you will create a copy of the part. If you are working on an assembly, you will create a new assembly file (but will not duplicate each of the parts in the assembly).

3. **Save As All** - Creates duplicates of all items in the current design. Save As All in an assembly will create a duplicate assembly file and duplicate part files.

14.2.2 Single Item Save vs. Multi-Item Save

When working with single items, and in some cases with assemblies and drawings, you may only be saving a single file. However, there are cases when multiple files will need to be saved simultaneously. Because Alibre Design allows you to design or change parts within the context of an assembly, and to change dimensions from within a drawing, there are several situations where you will be saving multiple files simultaneously.

The single file save dialog is the Windows Standard Save dialog. The multi-file save dialog is not and allows you to perform some common operations that make saving multiple files easier.

Examples of single file saves:

- Saving a part in a part workspace after modifying it.
- Saving an assembly after inserting some existing parts that have not changed since you inserted them.
- Saving a drawing after projecting a view.

Examples of multi-file saves:

- Saving a new assembly in which you have created new parts using top-down design. In this case, the assembly and all new parts must be saved simultaneously.
- Opening a drawing of an assembly and changing the driving dimensions of some items. In this case, the drawing, assembly, and affected part files will need to be saved simultaneously.
- Doing a **Save As All** operation. In this case, all items in the current design will be saved to a new copy simultaneously.

14.2.3 Saving Files to the File System

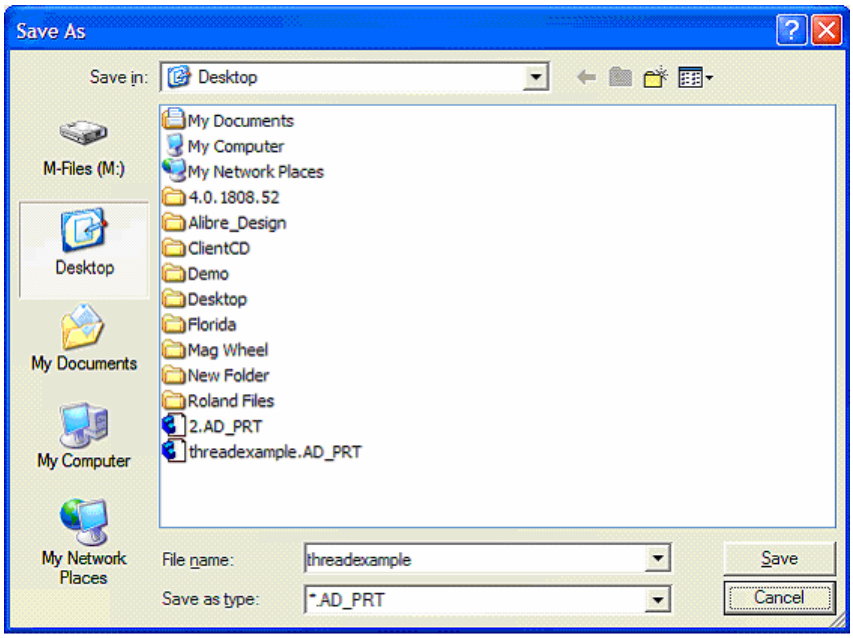
Alibre Design will determine for you whether multiple files must be saved simultaneously and will present you with the appropriate save dialog. Single item saves will use Windows Standard dialog. Multi-item saves will use a special save dialog.

➢ ***To save to the File System:***

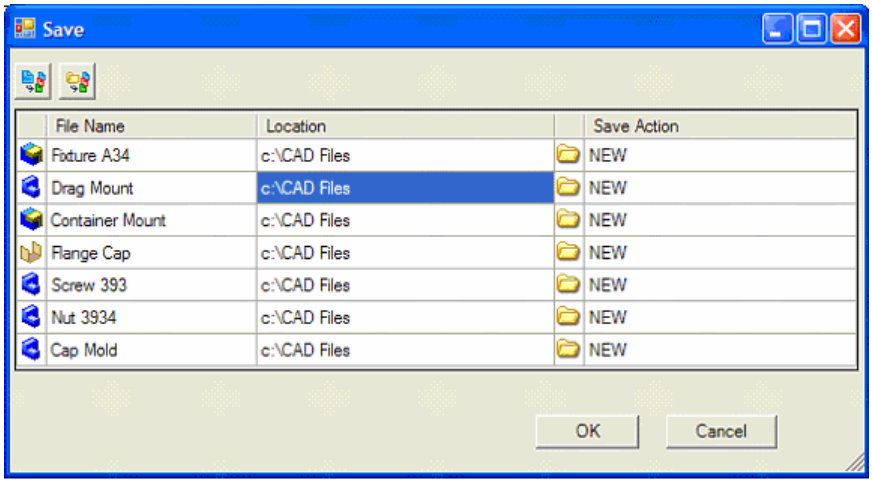
1. Determine which kind of save you want, based on the previous section.

2. Select the appropriate option from the File menu of the workspace that contains the items.

3. If the item has never been saved, or if you are doing a Save As or Save As all, you must name the file if it is a single item and press Save to finalize the save.



4. If multiple items must be saved simultaneously, you have some options. The multi-save dialog will display. In this case, we are saving a new top level assembly and some new parts we created in the context of the assembly. All of the items are new, and we will need to select names and locations for them. Previously saved items that may also need to be saved may also appear occasionally, depending on your workflow. You cannot change the name or location of previously saved items from this dialog.



You can manually set the location for each file if desired. There are also two tools available to assist you:

- - copies the location of the top level item (in this case, the main assembly) to all new files.
- - allows you to specify a location for all new files.

Note: Alibre Design will warn you if there are duplicate file names or any unsafe saving conditions. Hover over the exclamation points to learn about the warning.

5. Click **OK** to finalize the multi-item save.

14.2.4 Saving Files to the Alibre Vault

Saving to the Alibre Vault is similar to saving to the file system. The primary difference is, instead of entering a location for files as you would for the file system, you enter other types of data, such as the Product or Project to which the item belongs. The more data you define during a save that relates to the file you are saving, the more powerful the Alibre Vault becomes.

To activate saving to the Alibre Vault, please see *The Alibre Vault* (see "The Alibre Vault by M-Files" on page 347).

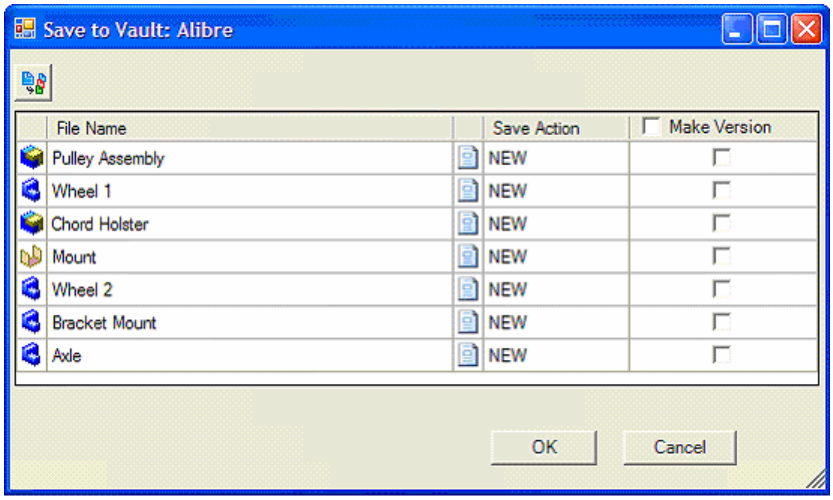
Note: Even if you have set the option to use the Alibre Vault, files will still be saved to the File System if they already exist on the File System. The Alibre Vault option in Alibre Design affects new files only. If you would like to save a file on the File System to the Alibre Vault, you will need to do a **Save As** or **Save As All** operation on the file.

➢ ***Saving items to the Alibre Vault***

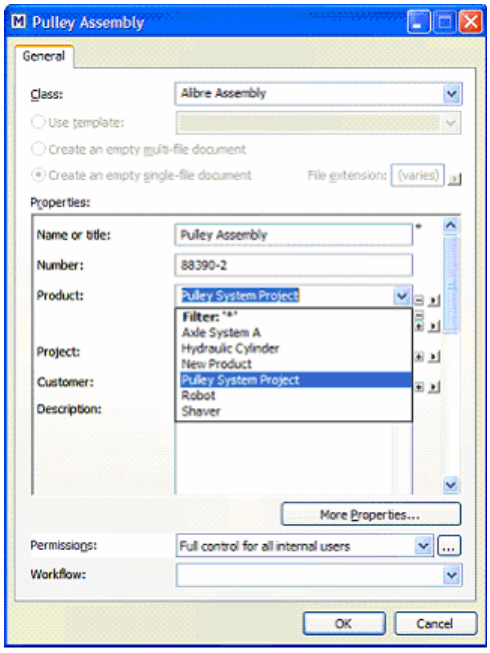
Unlike the File System, the Alibre Vault will always present you with a dialog during a save operation. This is because you are given the option to **Make Version**.

1. Select any Save option

2. The Single or Multi-Save dialog (shown below) appears.



3. You will notice there is no location field. Instead, you are given the option of opening the meta card for the file, which contains all the supplementary data you may use. Click the meta card button . The item's meta card appears.



4. Enter the data that you want to use for organizing, filtering, and searching this item. It is highly recommended that you always input either Product or Project data, as these are considered the primary filters. Do not change the Class data, located at the top. This is automatically populated.

5. Click **OK**.

6. Continue to set the metadata for each item in the list.

7. If you have several items that share similar data, such as Product, you can use this tool located at the top left of the dialog to copy meta data from one object to multiple objects. This is useful if you are saving a large assembly into the Alibre Vault for the first time and setting individual metadata for each file would be time prohibitive or error prone.

8. Select **Make Version** if you want to version the file. Versions are created every time a file is checked into the Alibre Vault. **Make Version** essentially checks the file in, and then immediately checks it back out so you can continue to edit it.

9. Click **OK** to finalize the save to the Alibre Vault.

CHAPTER 15

Importing and Exporting Data

Alibre Design's import and export functionality enables interaction with data from other CAD systems. Furthermore, machine tools and rapid prototyping can be driven from exported data created in Alibre Design. The STEP format is an ISO standard that is driven by industry. Alibre Design's native format is STEP AP 203 and AP 214, two protocols in the STEP standard. Alibre's STEP schema allows for the creation of parametric features along with geometry. This powerful capability and platform choice allows the data created in Alibre Design to be available for use beyond Alibre Design itself.

In This Chapter

Importing Data 305
Exporting Data 309
Special Options for IGES and STL Files 312

15.1 Importing Data

15.1.1 Supported File Types

A variety of file formats are supported for interoperating with data from other CAD systems. The import data types available are:

- **Solid Works Parts and Assemblies** (*.sldprt, *.sldasm)
- **STEP AP 203/214** (*.stp, *.step, *.ste)

  STEP (Standard for the Exchange of Product model data). An ASCII format set and driven by industry that all major CAD tools have adopted.
- **Alibre STEP** (*.stp)

  Feature-based STEP data inherent from Alibre's STEP schema.
- **SAT** (*.sat)

  ACIS file format.
- **IGES** (*.igs)

  Initial Graphics Exchange Specification. An ANSI-standard format.
- **DWG** (*.dwg)

  Standard file format for saving vector graphics from within AutoCAD.
- **DXF** (*.dxf)

  For drawing interchange format. An ASCII or binary file format of an AutoCAD drawing file for exporting AutoCAD drawings to other applications or for importing drawings from other applications.
- **3DM** (*.3dm)

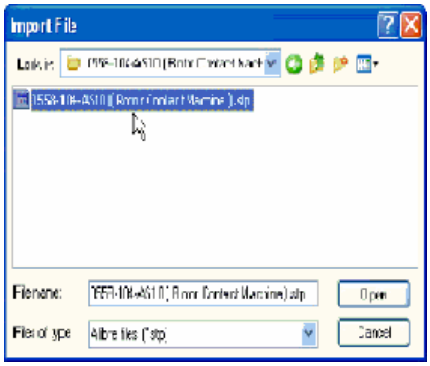
  Rhino Program file format.

15.1.2 Importing a File

You can import 3D parts and assemblies in **STEP**, **IGES**, or **SAT** formats. You can import 2D drawings in the **DXF** or **DWG** formats.

➢ ***To import a model file:***

1. Select the **Import** tool from the Standard toolbar; or from the **File** menu, select **Import**. The **Import File** dialog appears.



2. Browse to the location where the file is located and select the file.
3. Click **Open**.
4. If you have the option checked on, the **File Import Options** dialog appears if you are importing non-native Alibre STEP files, IGES files, and SAT files. See *Import Settings and Import Advisor* (on page 307) for more information.
   a. Select the applicable import options.
   b. If desired, you can uncheck **Show Options When Importing** and this dialog will not be displayed during subsequent file imports.
   c. Click **OK**.

Note: You can change whether or not the Import Options dialog is displayed during importing at any time. To do this, in a design workspace, choose **Tools > Options**. Select the Interoperability tab. In the Import File Options sections, check or uncheck **Show Options When Importing**.

The data appears in the workspace.

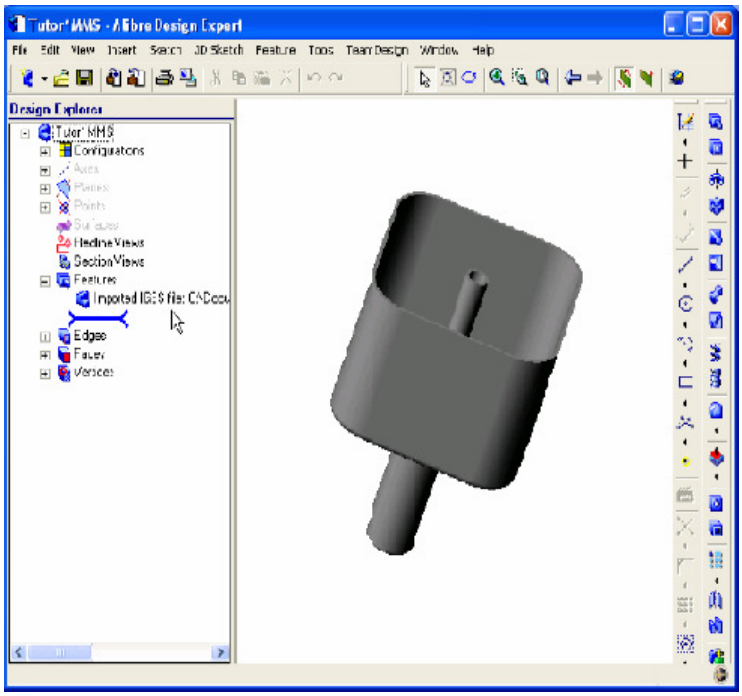


Figure 102: Part Workspace Displaying Imported IGES File

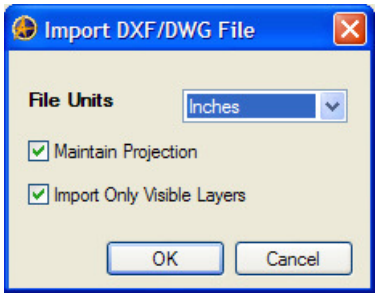
5. Note that the part is displayed in the Design Explorer with the label indicating the file type (e.g. **Imported STEP file** or **Imported SAT file**). Any feature added to the imported model will appear after this entry in the feature history tree.

➢ ***To import a DXF or DWG file:***

1. Select the **Import** tool from the Standard toolbar; or from the **File** menu select **Import**. The **Import File** dialog appears.

2. Browse to the location on the disk where the file is located and select the file.

3. Click **Open**. The **Import DXF/DWG File** dialog appears.



4. Select the **Units** for the imported file.

5. Check **Maintain Projection** to maintain the projection plane of the imported model in Alibre Design. If this option is unchecked, the model will be projected onto the XY-Plane in Alibre Design.

6. Check **Import Only Visible Layers** to bring in only the visible layers from the original file. Any "frozen" or "off" layers will not be imported. If this option is not checked, all layers will be imported. This can clutter the drawing and slow performance.

7. Click **OK**. The file opens in a drawing workspace.

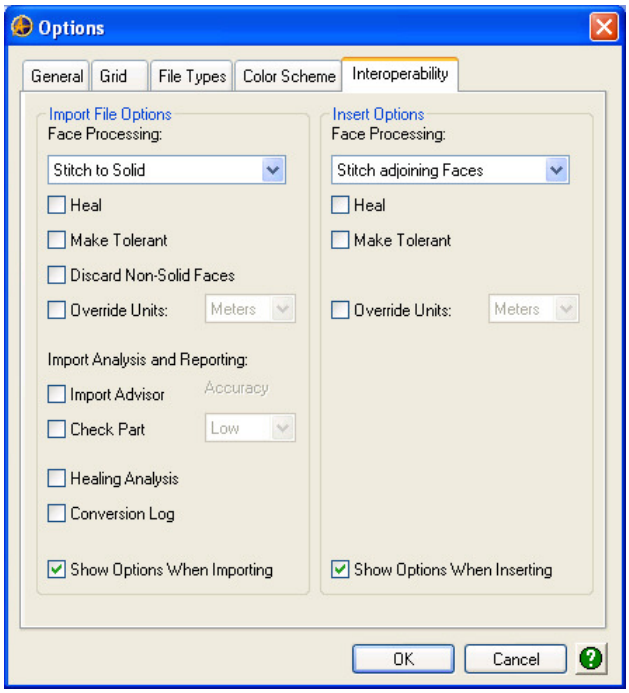
15.1.3 Import Settings and Import Advisor

Alibre Design's import settings provide various options to apply when importing data. The **Import Options** dialog will appear by default as you import data. You can also set default import options from any workspace.

➢ ***To set default import options from a workspace:***

1. From the **Tools** menu, select **Options.** The **Design Options** dialog appears.

2. Select the **Interoperability** tab.

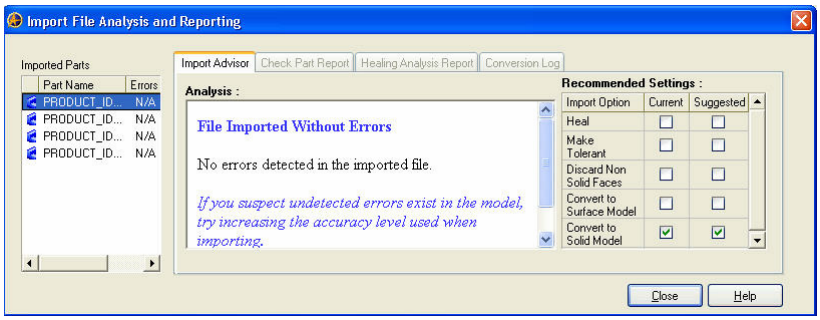


The Interoperability tab contains different selections that can be applied for imported data. The selections for import are as follows:

- **Heal**. Recalculates inaccurate geometry in order to make the part more accurate upon import.
- **Make Tolerant**. Tags inaccurate geometry for more intelligent subsequent operations.
- **Discard Non-Solid Faces**. Discards faces that are not part of a solid. These faces may have been created for reference.
- **Unstitch To Standalone Faces**. Converts a solid part to a set of faces. This can improve the visual representation of the part. This option is not recommended if changes will be made to the solid.
- **Stitch To Solid**. Converts a surface model to a solid part.
- **Override Units**. Converts the units to those specified.

Note: All import options can operate simultaneously, with the exception of **Convert To Surface Model** and **Convert To Solid Model**.

3. Select **Import Advisor** to generate an import summary after the file has been imported. With this option selected, a dialog appears after import displaying errors, suggestions and options that can be changed for the specific import issue.



4. Select the **Check Part** option to obtain information about the integrity of the file.

5. If **Import Advisor** or **Check Part** is checked, select an **Accuracy** level for the report:

   - **Low**: Fast error checks.
   - **Medium**: Slower error checks plus D-cubed curve and surface checks.
   - **High**: Slower warning and error checks plus D-cubed and surface checks.
   - **Very High**: Warning and error checks plus edge convexity change point and face/face intersection checks.

6. Select **Healing Analysis** if Heal is checked under Import Options. This produces a report on any data corrected as a result of the Heal command.

7. Select **Conversion Log** to view a report about import.

8. Check **Do Not Show Options When Importing** to bypass this dialog in the future. Options set in the Design Options dialog will be used automatically.

9. Check **Set As Default Import Options** to use these settings for future imports. Settings in the Design Options dialog will be updated.

10. If any Analysis and Reporting options are selected, the **Import File Analysis and Reporting** dialog appears.

Notes:

To save report data, highlight the text and press **Ctrl-C** to copy. The text can be pasted into another application for viewing or printing.

The tools that are available through the import options will provide a high degree of success in working with data from other CAD systems. For issues that are elusive, the Alibre Assistant should be contacted.

Options set in the Design Options dialog apply to all imports unless changes are made in the Import File Options dialog.

All import options can operate simultaneously, with the exception of Convert To Surface Model and Convert To Solid Model.

15.2 Exporting Data

You can export Alibre Design native data using a number of formats.

15.2.1 Supported File Types

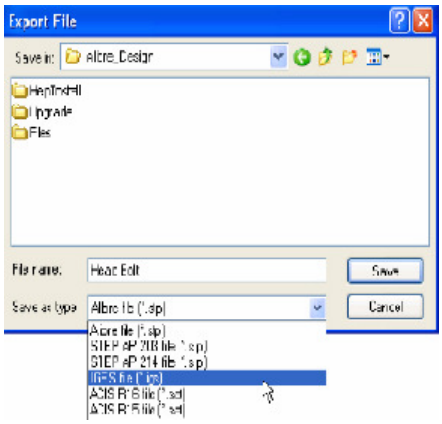
The table below summarizes the file types you can use to export data from Alibre Design.

Drawing	Alibre Design file (*.stp)	Image Files:
	AutoCAD DWG file (*.dwg)	JPEG Image file (*.jpg)
	AutoCAD DXF file (*.dxf)	Bitmap file (*.bmp)
	JPEG Image file (*.jpg)	PNG file (*.png)
	Bitmap file (*.bmp)	TIFF file (*.tiff)
	Enhanced Metafile file (*.emf)	GIF file (*.gif)
Assembly	Alibre Design file (*.stp)	Image Files:
	AP 203 file (*.stp)	JPEG Image file (*.jpg)
	AP 214 file (*.stp)	Bitmap file (*.bmp)
	ACIS 3.0-R10 files (*.sat)	PNG file (*.png)
	STL file (*.stl)	TIFF file (*.tiff)
		GIF file (*.gif)
Part	Alibre Design file (*.stp)	Image Files:
	AP 203 file (*.stp)	JPEG Image file (*.jpg)
	AP 214 file (*.stp)	Bitmap file (*.bmp)
	ACIS 3.0-R10 files (*.sat)	PNG file (*.png)
	IGES file (*.igs)	TIFF file (*.tiff)
	STL file (*.stl)	GIF file (*.gif)
BOM	CSV file (*.csv)	

15.2.2 Exporting a File

➢ ***To export a file:***

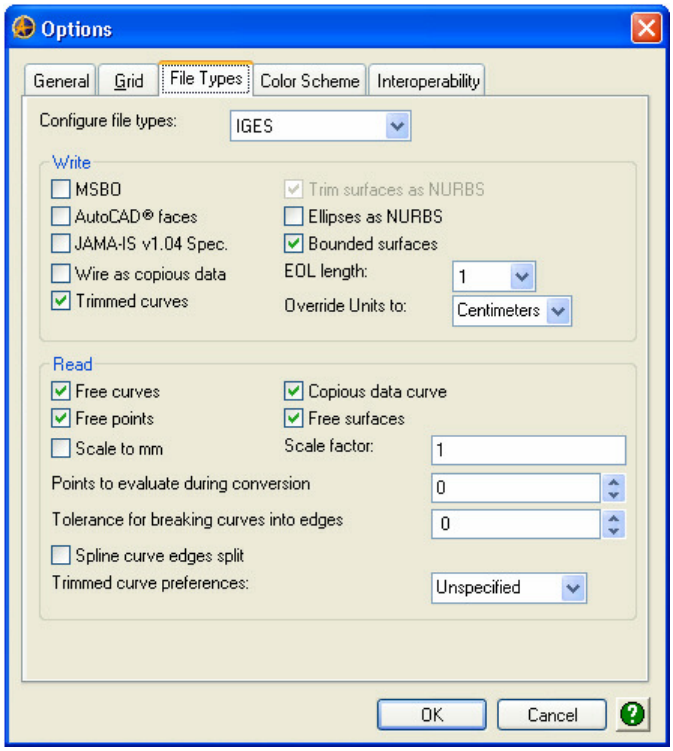
1. Select the **Export** tool from the Standard toolbar; or from the **File** menu, select **Export**. The **Export File** dialog appears.



2. Browse to the location where you want to export the file.
3. Specify the **File name**.
4. From the **Save as type** menu, select the appropriate file format to use.
5. Click **Save.**

15.3 Special Options for IGES and STL Files

Alibre Design has special option settings for **exporting IGES** and **STL** files and **importing IGES** files. These special settings are found in the **File Types** tab of the **Options** dialog. You can access the Options dialog by selecting **Options** from the **Tools** main menu in any 3D workspace.



CHAPTER 16

Alibre Motion

Alibre Motion™ is a motion simulation solution for analyzing the behavior of mechanical assemblies with moving parts. Using Alibre Motion, engineers and designers can create simulations and animations of systems that move, such as linkages, engines, automotive suspensions, conveyors and other mechanisms. It can be used to study and analyze how various components interact and behave according to engineering principles and physical laws.

With Alibre Motion you can build virtual prototypes to help analyze and optimize designs without having to build and test expensive and time-consuming physical prototypes. Alibre Motion enables multi-body dynamic analysis on an assembly created in Alibre Design, allowing you to realistically predict and visualize the assembly's motion.

Note: Alibre Motion is available in the Alibre Design Expert version. For more information on getting a license for Alibre Motion, please contact Alibre Sales.

Use Alibre Motion to:

- Create Virtual Prototypes with Automatic Constraint Mapping
- Quickly create animations that can be saved as AVI files
- Generate motion with prescribed rotations and translations
- Generate motions due to gravity and prescribed forces and torques
- Create accurate simulation models with a variety of physical elements
- Get simulation feedback from traces and X-Y result plots
- Detect interferences of overlapping bodies

Once installed and enabled, Alibre Motion is accessed from the main menu of the Assembly Workspace. The Motion Explorer, similar to the Design Explorer, allows you to view and interact with all of the Parts, Constraints, Physical Elements (Motors, Springs, Dampers and Motions) and Settings that make up a simulation.

In This Chapter

An Overview of Simulation 313
Installing and Enabling Alibre Motion 314
The Alibre Motion User Interface 314
Overview of Simulating and Playing 322
Forces and Torques in Simulations 322
Simulation Types and Parameters: 331
Results and Feedback from Alibre Motion Simulations 336
Detecting Interferences 344
Frequently Asked Questions (FAQ) 345

16.1 An Overview of Simulation

Newton’s laws of motion provide an accurate description of how bodies move under the influence of forces. However, except in the simplest of situations, applying those laws can be complex. The motion of a body can rarely be described by a single equation predicting its position and acceleration at any time in the future. Real designs usually involve multiple bodies and multiple forces or motions, all interacting with each other in ways that are too complex to solve without the aid of computer simulation. The field of physics and mathematics involved in these calculations is known as Rigid Multi-Body Dynamics.

What does a Simulator do?

In reality, time moves smoothly. A simulator, in contrast, needs to break time into a series of calculation time-steps, or intervals, much as a movie or animation is broken down into a series of frames. Starting from a set of initial conditions at the beginning of a time-step, the new positions and accelerations of each body in the system are calculated using Newton's laws from the forces and torques acting on them, and what is known about their physical characteristics.

What Makes up a Simulation?

- The type of simulation to perform and its driving parameters
- The physical characteristics of the bodies that make up the system
- The constraints that dictate the way the bodies influence each other
- The forces and torques acting on or within the system
- Any prescribed motions to be forced upon the system

16.2 Installing and Enabling Alibre Motion

Installing Alibre Motion

Before beginning to use Alibre Motion, you must first install the Alibre Motion Add-On.

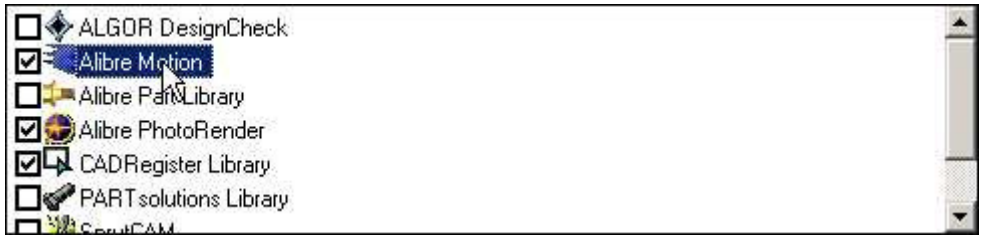
Note: You must have a license for Alibre Motion to install and enable it. Alibre Motion is available for the Alibre Design Expert version.

Enabling Motion

Once you have installed the Alibre Motion Add-On, you can enable Motion from the Home window or an Assembly Workspace.

➢ ***To enable Alibre Motion:***

1. From the **Tools** menu, select **Add-On Manager**.
2. Find Alibre Motion in the list of available Add-Ons and check the box beside it.
3. Alibre Motion loads immediately. You will notice that there is now a **Motion** main menu item in the Assembly Workspace.



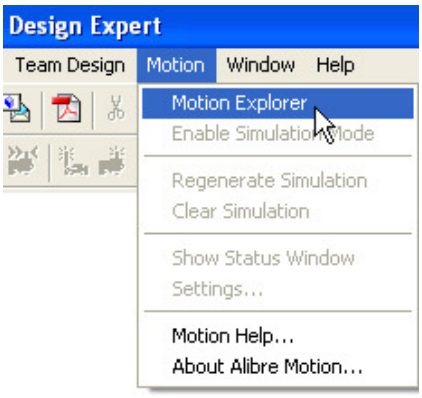
16.3 The Alibre Motion User Interface

Once installed and enabled, Alibre Motion is accessed from the main menu in an Assembly Workspace. The three main components of Alibre Motion are:

- The Motion main menu: allows you to enter Alibre Motion when you are in an assembly workspace.
- The Motion Explorer: similar to the Design Explorer, allows you to view and interact with all the Parts, Constraints, Physical Elements (Motors, Springs, Dampers and Motions), and Settings that make up a Simulation.
- The Playback Deck: allows you to start the generation of a simulation and control playback of the animation.

16.3.1 The Main Alibre Motion Menu

The Motion main menu can be accessed in an Assembly workspace when Motion is installed and enabled.



Motion Explorer

This activates Alibre Motion for the current Assembly. When this is selected, the Motion Explorer is shown, where you can add and change any physical elements that you want to use in your simulation.

Enable Simulation Mode

When you have added all of the desired elements, this puts the Assembly workspace into Simulation/Playback mode. In this state, all the constraints in the Assembly are suppressed, and any Anchored bodies are freed. The Simulator takes control of the placement of all moving parts and sub-assemblies, in order to animate them as Simulation frames are generated.

Regenerate Simulation

When you have made changes to your Assembly, use this to update the Simulator. You may be prompted to do this occasionally if you have made a change that renders the Simulator unable to animate the model.

Clear Simulation

This completely resets the Assembly design to create a blank simulation.

Note: All changes you have made to the Simulation set-up, physical elements you have added, and feedback elements you have defined, will be lost.

Show Status Window

The Status window shows any Warnings and Simulator messages generated when checking the simulation model, or generating a simulation. If you have closed the window, it can be re-opened by selecting this item.

Settings

This item takes you to the Settings dialog, where you can specify the parameters that determine how the Simulation should be generated and played back. From here you can also specify how Traces appear, and other options.

16.3.2 Alibre Motion Explorer

Much like the Design Explorer, the Motion Explorer shows all the items which make-up a Simulation set-up. From here, using the right-click menu on headings and individual items, you can create new Simulations, insert Physical Elements, specify measurements that should be taken, and Traces to be shown.

➢ ***To launch the Motion Explorer***

1. In an assembly workspace, from the Motion main menu, select Motion Explorer.

2. The Motion Explorer opens, and either loads existing Simulation Setup Data if the open Assembly file contains it, or automatically generates a new Simulation, reading in all the physical data and characteristics of the parts, and constraints that make up the assembly.

Figure 103: Motion Explorer

Automatic Simulation Creation

The first four group headings (Configuration, Fixed Parts, Moving Parts, and Constraints) contain all of the Simulation elements that Alibre Motion creates automatically for you by analyzing your Assembly design, using *Automatic Body Mapping* (see "Moving and Fixed Parts" on page 333) and *Automatic Constraint Mapping* (see "Automatic Constraint Mapping (ACM) in Alibre Motion" on page 333).

Additional Simulation Elements

The remaining group headings contain those elements you may choose to add to create your Simulations, and to define any feedback and results you may require.

16.3.3 Motion Explorer Groups

Configuration and Simulations

The currently active Assembly Configuration is shown with all existing simulations based on it. In the Motion Explorer image in the previous section, the default configuration, “Config<1>” is active, and there is only one Simulation, “Simulation<1>”, which is active.

Fixed Parts

Those parts that Alibre Motion has determined cannot move during the simulation are shown under this heading. These are anchored Parts and those anchored Sub-Assemblies that are not “Flexible." These do not affect the dynamics of the simulation, but may hold the points to which constrained Moving Parts attach. Under each Fixed Part is a Constraints sub-heading listing all attached constraints. This duplicates a portion of the Constraints list.

Moving Parts

These are the dynamic Parts that have the potential to move during the simulation. Each Moving Part is shown with a Part icon or a Sub-Assembly icon depending on which it represents. Under each Moving Part is a Constraints sub-heading listing all attached constraints. This duplicates a portion of the Constraints list.

Note: This list of Moving Parts will not necessarily be the same as the Parts and Sub-Assemblies in the top level of the Design Explorer. For instance, if a Sub-Assembly is made "Flexible," there will be Moving Parts that come from that Sub-Assembly.

Constraints

This contains a list of all the constraints that the Automatic Constraint Mapping has determined should take part in the Dynamic Simulation. If a constraint is suppressed at the time when the Simulation is Created or Regenerated, it is not used by the simulator.

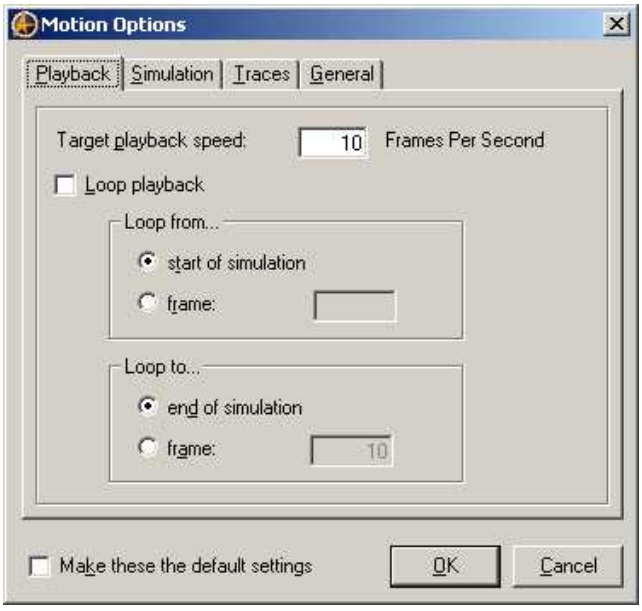
Note: The Automatic Constraint Manager creates a Motion Equivalent Constraint for almost all permissible Alibre Constraints. If a constraint cannot be used by Alibre Motion, the icon is shown with a question mark, and is excluded from Simulations. If the constraint is important to the Assembly, you need to try to constrain it differently in your assembly, then regenerate the Simulation.

16.3.4 Motion Settings

Settings for Alibre Motion can be specified by choosing **Settings** from the **Motion** main menu.

Playback Tab

Settings on the Playback tab are stored in the assembly. You can specify different settings for each Simulation in the assembly.



➢ ***To set Playback options:***

1. In **Target playback speed**, specify the maximum speed at which Motion should animate the simulated frames.

   Note that the actual playback speed that can be achieved is dependent on many factors, particularly the speed of your system and its graphics capabilities, and the complexity of the model being simulated. See Optimizing Playback Performance for more information.

2. Check the **Loop playback** checkbox if you wish to loop between the loop-start and loop-end points. This is also achieved by pressing the Loop button on the Playback Deck.

3. In **Loop from**, choose either:

a. **start of simulation** to loop to the first simulation frame when playback reaches the loop-end point; or,

b. **frame** to loop to the specified frame number when playback reaches the loop-end point.

4. In **Loop to**, choose either:

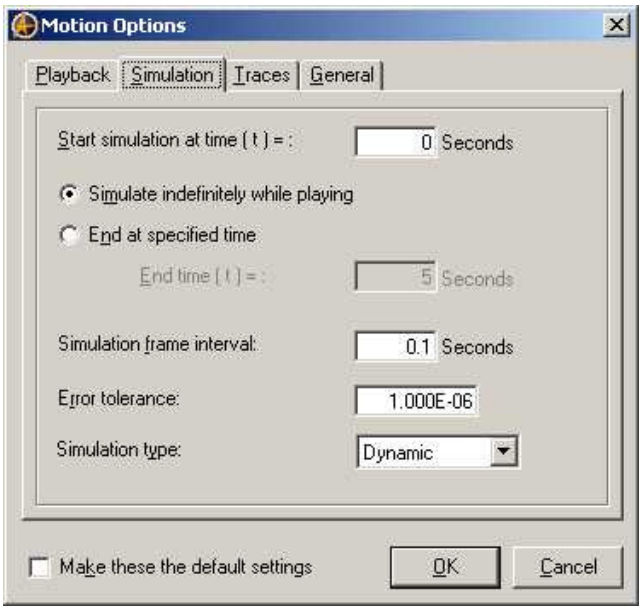
a. **end of simulation** to continue the playback all the way to the final simulation frame available; or,

b. **frame** to loop to the loop-start when it reaches the specified frame number.

Note: If you change the Playback settings after a simulation has been started, and the current frame of the simulation is already past the frame you set as your Loop to frame, the playback will continue on without looping.

Simulation Tab

Settings on the Simulation tab are stored in the assembly. You can specify different settings for each Simulation in the assembly.



➢ ***To set Simulation Options:***

1. In **Start simulation at time (t) =**, specify the time (t) at which the Simulation should start. The variable “t” is used internally by the Simulator in many of the equations that define the Physical Elements used in simulation, including Harmonic (sinusoidally varying) functions.

2. Choose one of the following options:

a. **Simulate indefinitely while playing**: When selected, the Simulator will continue generating frames as long as you are in Play mode. When the current frame being shown reaches the last available simulation frame, the Simulator is restarted, and each frame is shown as it is generated.

Note: The maximum number of frames that Alibre Motion can generate for a single simulation is 32,767.

b. **End at specified time**: Specify the maximum amount of time in seconds that should be simulated. When the Simulator reaches this point, it is stopped.

3. In **Simulation frame interval**, specify the output time-step or simulation frame interval. The Simulator produces an output frame after each interval. If the Target Playback Speed matches this interval, and your model and system allow playback at that speed, you can produce a Simulation playback in real-time.

Note: The Simulator internally determines the time-step required to produce an accurate simulation, and this may vary throughout the simulation. However, it only produces an output frame for animation at intervals as specified by this parameter.

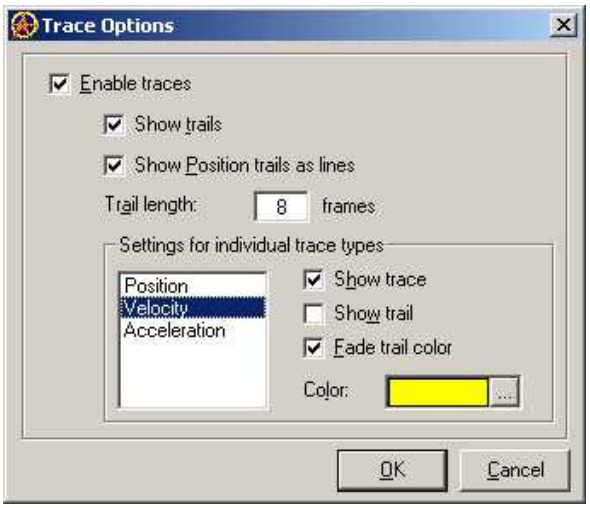
4. In **Error Tolerance**, specify the error tolerance allowed internally by the Simulator during calculation. The allowable range of values varies, depending on the type of simulation specified in “Simulation Type." Under some circumstances, lower values may impact Simulation performance, and the highest value is selected by default.

5. In **Simulation type**, specify the type of Simulation to be performed. Choose the desired type from the drop-down list. See *Simulation Types and Parameters* (see "Simulation Types and Parameters:" on page 331) for more information.

Traces Tab

Note: This dialog is available on the Settings dialog, or by selecting **Trace Options** from the right-click menu of any Trace listed in the Motion Explorer.

This form is used to specify the type, length, and default colors of Traces shown during playback to visualize the paths of Parts and Sub-Assemblies, and to get feedback concerning their Velocities and Accelerations.



➢ ***To set Trace options:***

1. Check **Enable traces** to show traces during playback. No traces of any kind will be shown during playback unless this box is checked.

2. Check **Show trails** to see indications from previous animation frames. These trails can be extremely useful for visualizing the paths taken by Parts. If no Individual Trace Types have the Show Trail option selected, or no Traces are currently being generated, no trails will be shown.

   Note: Trails are always shown stretching backwards in time from the current frame, even during Reverse playback.

3. Check **Show Position trails as lines** to show the Position indicator for previous frames as a series of straight lines joining each previous Position. If this box is NOT checked, previous Positions are shown as a series of discreet Position indicators.

4. In **Trail length**, specify the maximum number of previous frames for which to show trails. Note that during the first frames of animation, when the current frame number is less than this figure, trails will be shorter than this number. Specifying extremely long trails may degrade playback performance.

5. In **Settings for individual trace types**, you may specify how individual trace-types are displayed. Select in the list-box the trace-type you want to change, and for that type you may specify:

   a. Check **Show trace** to see visual indicators at the current frame during animation playback.

   b. Check **Show trail** to see visual indicators for previous frames during animation playback.

   c. Check **Fade trail color** to see visual indicators for previous frames fading to black over the length of the trail.

   d. In **Color**, set the color of the visual indicator for the selected trace type. To select a different color, press the Browse button: .

General Tab

➢ ***To set General options:***

1. Check Show all warning, information and confirmation dialogs to see pop-up dialogs with messages and warnings while working in Alibre Motion.

2. Check Show Simulation Messages to see messages concerning the actions and status of the Simulator in the Motion Status Window.

3. Check Save Simulation Data in Assembly File if you want to save your simulation data with the assembly. If you do NOT want your simulation information saved, un-check the box.

Note: If this box is not checked, any Simulation setup data, actuators, springs, dampers and feedback elements will not be saved, and will be lost when you close the Assembly workspace.

When you have finished setting your options on all of the tabs in the Motion Options dialog:

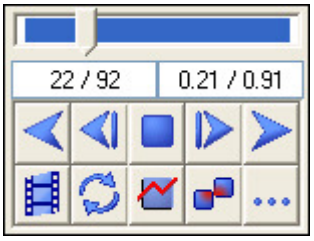
1. Check **Make these the default settings** to save these settings as the default for all new Simulations generated from that point on. (This option does not apply to the General Tab, so it will be grayed out if you have that tab selected. Select any other tab and then check the option on.)

2. Click **OK** to apply your changes and exit the dialog, or click **Cancel** to abandon your changes and exit the dialog.

16.3.5 Playback Deck

This section of the Motion Explorer is used to control the starting and playback of Simulations. From here you also control the X-Y Plots dialog, the Interferences dialog, the Video Generation (AVI) dialog, and the Settings/Options dialog.

Note: The Playback deck is only enabled when you are in Simulation Mode. Select Simulation Mode from the Motion main menu.



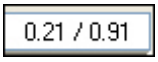
Frames Slider

Gives a graphic indication of the number of simulated frames available for playback, and the current frame. To jump to another frame, either drag the slider, or click on the background on either side of it. Whenever the slider is positioned at the far right during playback, the Simulator will be running, generating more Simulation frames.

Current-Frame / Total Frames

Shows the current frame, and the total number of simulated frames.

Current-Time / Total Time



Shows the time (t) of the current frame, and the total simulated time available.

Note: Control the simulation frame interval (step size) in the Simulations Settings Tab of the Settings form.

Button	Name	What it does
	Play	Starts the simulation and animation process.
	Step Forward	Steps forward by one frame.
During playback	Pause / Stop	During playback, stops the animation at the current frame. When paused, resets to the first simulation frame.
When paused		
	Step Back	Steps backward by one frame.
	Play Backwards	Plays the simulation backwards from the current position.
	Video Generation (AVI)	Opens the Video Generation (AVI) form to enable you to generate movies of your simulations. See Generating Video for more information.
	Loop Playback	Loops the playback between specified start and end points.
	X-Y Plot	Opens / Closes the X-Y Plot
	Interferences / Collisions	Opens / Closes the Interferences form
	Settings / Options	Opens the Settings form, where you can specify Simulation, Playback, Trace and other settings and options.

16.4 Overview of Simulating and Playing

Once you have Alibre Motion installed and enabled, there are simple steps to follow to create and animate a Simulation from your Assembly design.

The Basic Steps

1. Activate the Motion Explorer: Select **Motion Explorer** from the **Motion** main menu.
2. Add Physical Elements: right-click on constraints and other headings in the Motion Explorer, enable gravity, and add any motors, actuators, springs, and dampers required.
3. Enter Simulation Mode: Select **Enable Simulation Mode** from the **Motion** main menu.
4. Press Play: Press the **Play** button on the Playback deck

In addition you may want to…

1. Show a Trace of the position of a moving part: Right-click a Moving Part and select **Trace**, then select a quantity to trace.
2. Create a Video of the animation: Press the **Video** button on the Playback deck.
3. Show an X-Y Plot of the position, velocity or another dynamic measurement of a moving part: Right-Click a moving part and select **Dynamic Measurement**. Make sure **Show In XY Plot** is checked, and after specifying the measurement, press the X-Y Plot button on the Playback deck.
4. Make changes to your design and regenerate the Simulation: Right-click the Simulation item on the Motion Explorer, and select **Regenerate**, or select **Regenerate** from the **Motion** main menu.
5. Detect Interferences: Press the **Interferences** button on the Playback deck.
6. Export dynamic measurements for analysis: Right-Click a Part and select **Dynamic Measurement**. After specifying the measurement, right-click that Measurement item in the Motion Explorer then select **Export**.

16.5 Forces and Torques in Simulations

Within any but the simplest of mechanical systems there are numerous forces and torques acting on each body within the system. Some are specified by the person who created the simulation model, and usually many more are a result of the interactions of the constrained bodies according to Newton's third law. Newton's third law states that for every action there is an equal and opposite reaction, and is encapsulated in physics terms by the laws of Conservation of Momentum, and Conservation of Angular Momentum.

Forces External to the System

The law of Conservation of Momentum states that in the absence of any external forces, the total momentum of a system is conserved. For simulation purposes, the only ways to add external forces or torques to a system are:

- Enabling Gravity: When it is enabled, gravity acts on the entire system, and a constant magnitude and direction is used throughout (you can specify the components of gravity, or choose the default value of 9.81 m/s^2 in the direction of the negative Y-Axis).
- Constraining a Part to a “Fixed” or “Anchored” Part, or a Geometry Feature (Planes, Axes) within the main Assembly.

Forces Internal to the System

Internal forces can change the configuration of your assembled designs, but on their own they cannot influence the total momentum, either linear or angular, of the system.

Constraints between “Moving Parts” can be considered as entirely internal to the system.

Note: You don't need to explicitly distinguish between these in any way - Alibre Motion will simulate the Assembly as you have designed it.

Physical Elements: Motors and Actuators, Springs and Dampers

One step in the process of producing a simulation is specifying any physical elements that transmit, produce, or modify forces and torques. The Assembly constraints within your designs specify where physical elements can be placed, and there may be times that you add constraints to your design solely for the purpose of adding a physical element such as a spring. An enormous number of real-world interactions can be accurately modeled by using combinations of these. Depending on the type of constraint and the details of its geometry, you may be able to add some or all of the following:

- Motor (or Rotary Actuator): Produces a torque or rotational motion. For instance, a car engine, a winch, or a DVD player motor.
- Rotary Spring: Produces a torque opposing rotational displacement. Real-world equivalents include a door hinge-spring and a coiled watch-spring.
- Rotary Damper: Produces a torque opposing rotational motion, for instance a car clutch, or an electric generator. They can also be used to model "wet" friction.
- Linear Actuator: Produces a force or motion in a particular direction, for instance an hydraulic ram, or an electric car-window lifter.
- Linear Spring: Produces a force which acts to oppose displacement in a straight line, such as an elastic band, or a car suspension spring.
- Linear Damper: Produces a force which acts to oppose motion in a straight line, such as a shock-absorber. Can also model "wet" or sliding friction.

You can expect nothing to happen unless at least one physical element has been added. In some circumstances, enabling gravity may be all that you require, but more often, you will need to add the motors, springs and dampers found in the mechanism you are designing.

16.5.1 Adding Physical Elements (Motors and Actuators, Springs and Dampers)

➢ ***To Add a Physical Element***

1. Right-click on a Constraint in the Motion Explorer.
2. Choose **Insert** from the pop-up menu.
3. Select from the enabled options:

Note: If you have already added a Physical Element of a particular type to that constraint, the option will be shown disabled.

A dialog opens to allow you to specify the setting for the element you are adding.

Linear or Rotary

All moving parts in your simulations are free to move in any way, except where specifically constrained. When simulated, they act under the influence of constraints, gravity, and forces, torques, and motions specified by you, using physical elements. The types of physical element available in Alibre Motion are divided into two broad categories.

- Rotary - concerns torques and motion around a central axis
- Linear - concerns forces and motion in straight lines

Constraints Define Types of Physical Elements Available

The Assembly Constraints and their geometries and settings defined in your designs dictate the type of physical elements you can add to create simulations. The types of motions allowed by the constraint in Alibre Design can give you a good idea of the types of physical elements that are available for that constraint, and shown in the "Insert" sub-menu.

For instance, an “Align” constraint between two straight edges allows rotation of each part around the lines, as well as linear motion along the lines. When mapped into Alibre Motion, you are able to add both Rotary elements and Linear Elements to this type of constraint.

Constraints Define Attachment Points of Physical Elements

Any physical element produces, transmits or modifies a force or torque between two “Attachment Points,” or points of action. These points are taken directly from the features used to create the Constraint in your Assembly.

16.5.2 Actuators (Motors and Linear Actuators)

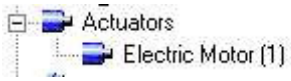
When you add a Motor or Actuator to your Simulation, they may be specified in either one of two ways:

For Motors (Rotary Actuators)

- Torque type
- Prescribed Motion (Rotation) type

For Linear Actuators

- Force type
- Prescribed Motion (Translation) type



Each type has its own characteristics and uses:

- Force-Torque Actuators - These actuator types are used to specify the force (linear) or torque (rotary) acting on the connected parts. Depending on the physical characteristics of the parts, and any other parts and physical elements connected to them, these actuators may or may not produce any resultant motion, in the same way that pushing on a door does not necessarily open it.
- Prescribed Motion Actuators - These actuator types are used to specify the actual motion of the connected parts, disregarding the force or torque that that motion would require.

➢ ***To Add an Actuator:***

1. In the Motion Explorer, right-click on a constraint.
2. Select **Insert**, then select either **Linear** or **Rotary Actuator**. The Actuator Settings dialog opens.

Note: Depending on the constraint, you may be able to add one, both, or neither type of Actuator.

3. In **Type**, choose either a Force motor or a Linear Motion motor. (For Rotary Actuators (motors), you may choose between a Torque motor, and a Rotation motor.)
4. Choose either **Constant** or **Harmonic** function

➢ ***To Edit an Actuator:***

Right-click the Actuator in the Motion Explorer, and select **Edit.**

Or

Double-click the Actuator item.

➢ ***To Re-Name, Delete, or Suppress an Actuator:***

Right-click the Actuator in the Motion Explorer, and select the required option. Suppressed Actuators are shown grayed out.

Force- and Torque- Type Actuators

In many circumstances, simulations using these types can be the more realistic choice. Actuators of these types may be opposed by springs, dampers, and the inertias of any driven parts in a realistic way.

Forces and Torques can be defined in the following ways:

- Constant Force or Torque
- Harmonic (or Sinusoidally varying) Force or Torque

Prescribed Motions and Rotations

Actuators of the Prescribed-Motion type produce the specified motion of the connected parts, no matter what magnitude of force or torque would be required to do so in the real world. They are very useful for answering sizing questions like: *"Assuming this part is to rotate at 5rpm, what torque would be required?"*, and are often used for modeling sub-sections of larger systems where the required output of the system is specified in advance, and for examining the ranges of motion of parts.

Prescribed Motions can be defined in the following ways:

- Constant Motion
- Harmonic Motion (or Sinusoidal Motion)

Some characteristics of Prescribed Motions and Rotations

- Prescribed Motion Actuators are Incompatible With Springs and Dampers

  Since Springs and Dampers produce force or torque effects, any Prescribed Motions acting on the same constraints will completely override them. In this situation, any Springs or Dampers will be automatically excluded from the simulation (Springs and Dampers on other constraints remain unaffected, and their effects will be calculated normally).

- Prescribed Motion Actuators Can Produce Locking

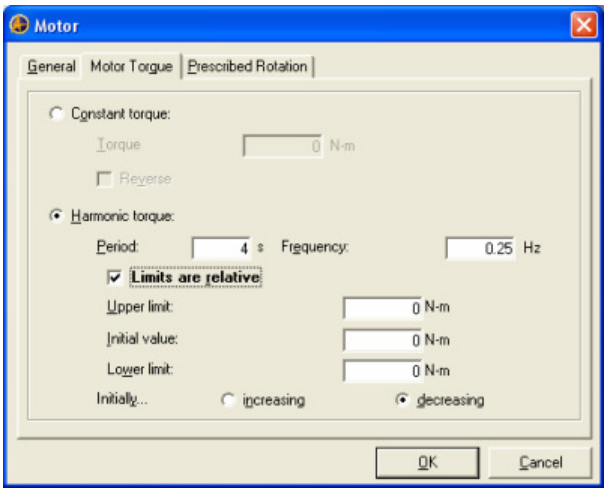
  If you prescribe a Motion that cannot continue past a certain point due to other constraints, you have created a situation where an "irresistible force meets an immovable object," as the Simulator does not decide which constraint or motion should be ignored at that point. Metaphysical considerations aside, this produces a simulation condition for which there is no viable solution, so the simulation stops at that point, and a warning message is displayed.

- Quasi-Kinematic Simulations Always Use Prescribed Motion Types

  The inputs to Quasi-Kinematic simulations that you create are always prescribed motions, as forces, torques and inertias are effectively ignored in this type of simulation. See Simulation Types and Parameters for more on Quasi-Kinematic simulations. You can use prescribed motions in Dynamic Simulations, as well.

Specifying Harmonic Functions

Various quantities in Alibre Motion, including prescribed motions, forces, and torques, may be specified as Harmonic or Sinusoidally varying functions of time. This enables you to create back-and-forth motions, for instance.



To completely specify a Sinusoidal function, three things must be known:

- **Period** - The Period is the time taken for one complete cycle. You can enter the Period in seconds, which is the time taken for one complete cycle, or the Frequency in Hertz, which is the number of cycles per second. Entering one automatically calculates the other.
- **Amplitude** - The Amplitude of a Sinusoidal function is defined as the difference between the mid-point and one extreme or the other. In most applications, we know the range of motion or force required, so Alibre Motion allows you to specify the Amplitude by entering the Upper and Lower limits.
- **Phase** - The Phase is most easily thought of as the position within a cycle. In Math and Engineering, Phase is usually specified in Radians or Degrees, but since this may often not easily translate to the problem at hand, Alibre Motion specifies the Phase as a combination of the Initial value and whether the value is initially increasing or decreasing. For instance, if the Initial Value is half way between the Upper and Lower limits and increasing, this would represent a Phase of 0. If it is initially decreasing, the Phase would be 180 degrees, or Pi Radians.

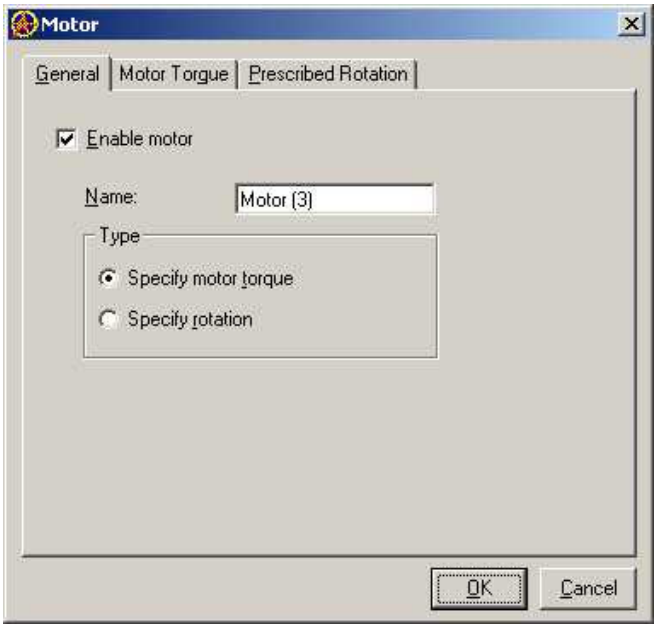
When you first start a Simulation, the positions of the parts, either Linear or Angular, are taken from your Assembly Design. If you are specifying a Harmonic Prescribed Motion, you will see that the Initial Value box is disabled. In this case, if you want to specify different initial conditions, ensure that you are not in Simulation Mode, change the position of the Part, and regenerate the Simulation.

Often you will know the range of motion or force relative to the initial value, rather than as absolutes. Alibre Motion allows you to specify the Limits in either absolute or relative terms. To specify that the values you enter should be interpreted as relative to the Initial Value, check the "Limits are relative" check-box.

Actuator Settings

General Tab

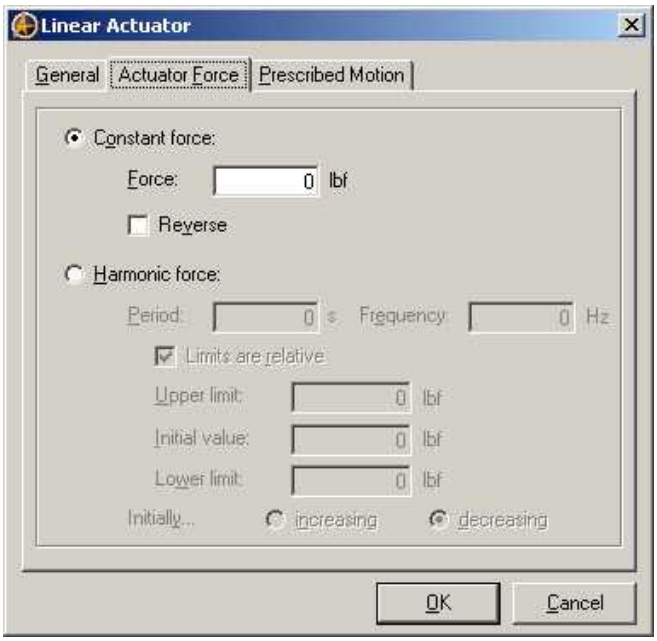
Note: Applying changes to the values in this dialog will reset the current simulation.



1. Check **Enable motor** (or Enable actuator) to enable this actuator, or uncheck to disable it.
2. In **Name**, specify the name of the actuator as it should be shown on the Motion Explorer, and in any Dynamic Measurements.
3. In **Type**, choose either:
   a. Specify motor torque (or Specify actuation force) for a motor that produces a force or a torque.
   b. Specify rotation (or Specify motion) for a motor that produces a prescribed motion.

Actuator Force Tab

Note: Applying changes to the values in this dialog will reset the current simulation.

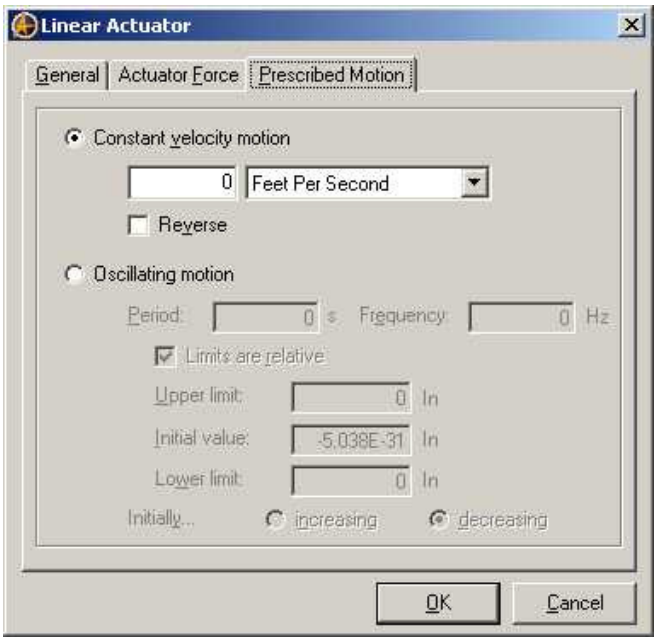


Select either **Constant force (or Constant Torque)** or **Harmonic force (or Harmonic Torque)**

- Constant torque / force - specifies that the Torque or Force produced does not vary over time.
  - In **Force** (Torque), enter the force to be produced by the actuator. Note the units displayed.
  - Check **Reverse** to reverse the sign of the force (torque).
- Harmonic force / Harmonic torque - specifies that the force or torque produced by the actuator varies sinusoidally over time.

Prescribed Motion Tab

Note: Applying changes to the values in this dialog will reset the current simulation.



Select either **Constant velocity motion (or Constant rotation)** or **Oscillating motion (or Rotary Oscillation)**.

If you select Constant velocity motion, the motion produced does not vary over time.

1. Specify the velocity or angular velocity to be produced by this actuator, then select the units from the drop-down list.

2. Check **Reverse** to reverse the direction of the motion.

If you select Oscillating motion, the displacement produced by the actuator varies sinusoidally over time.

When you have finished making changes on each of the tabs, choose **OK** to save and apply the changes you made and close the dialog; or choose **Cancel** to abandon changes and close the dialog.

16.5.3 Springs

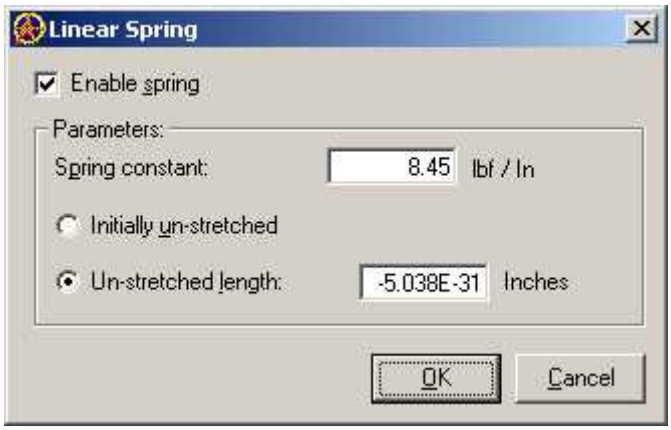
Springs in Alibre Motion are assumed to perfectly obey Hooke's law at all times, and have no non-linear behavior.

➢ ***To Add a Spring***

1. In the Motion Explorer, right-click on a constraint

2. Select **Insert**, then select either **Linear** or **Rotary spring**. The Spring Settings dialog appears. To define a Spring, you must specify the Spring constant, which is the force or torque produced per unit of linear or angular displacement from the unstretched position. The force produced by a Spring always opposes the displacement.

Note: Depending on the constraint, you may be able to add one, both, or neither type of Spring.



Note: Applying changes to the values in this form will reset the current simulation.

3. Check **Enable spring** to enable this spring, or uncheck to disable it.

4. In **Spring constant**, enter the Spring Coefficient, or Spring Constant. Note the units shown to the right of the text-box.

5. Select **Initially un-stretched** if you want the spring assumed to be initially unstretched in the design at the start of simulation. Any subsequent deflection will cause the spring to produce a force or torque acting towards this initial point.

6. Select **Unstretched Length** / **Unstretched Angle** if you want the spring assumed to be already stretched in the original design, at the start of animation. You can specify the natural length or angle of the spring in the text-box to the right.

7. Select **OK** to save and apply the changes you made; or click **Cancel** to abandon your changes.

➢ ***To Edit a Spring***

Right-click the Spring in the Motion Explorer, and select **Edit**, or simply double-click on the spring item.

➢ ***To Re-Name, Delete, or Suppress Springs***

Right-click the Spring in the Motion Explorer, and select the required option. Suppressed Springs are shown grayed out.

16.5.4 Dampers

Dampers in Alibre Motion have no non-linear behavior, and produce a force or torque that is perfectly proportional to the linear or angular velocity. The force or torque produced by a damper always opposes the motion.

➢ ***To Add a Damper:***

1. In the Motion Explorer, right-click on a constraint, and select **Insert**

2. Choose either **Linear** or **Rotary damper**. The Damper dialog opens.

Note: Depending on the constraint, you may be able to add one, both, or neither type of Damper.

3. Check **Enable damper** to enable this damper, or uncheck to disable it.
4. In **Damping coefficient**, enter the coefficient of damping to be applied at the constraint. Note the units shown to the right of the textbox.
5. Click **OK** to save and apply changes; or click **Cancel** to abandon changes.

➢ ***To Edit a Damper:***

Right-click the Damper in the Motion Explorer, and select **Edit**

OR

Double-click the Damper item.

➢ ***Re-Naming, Deleting, and Suppressing Dampers***

Right-click the Damper in the Motion Explorer, and select the required option. Suppressed Dampers are shown grayed out.

16.5.5 Gravity

By default, the acceleration due to gravity (often approximated as 9.81 m/s^2, or 32.185 ft/s^2 at the Earth's surface) is NOT enabled in Alibre Motion. You can enable gravity and change the values and direction used by the Alibre Motion simulator from the Motion Explorer.

➢ ***To Enable Gravity:***

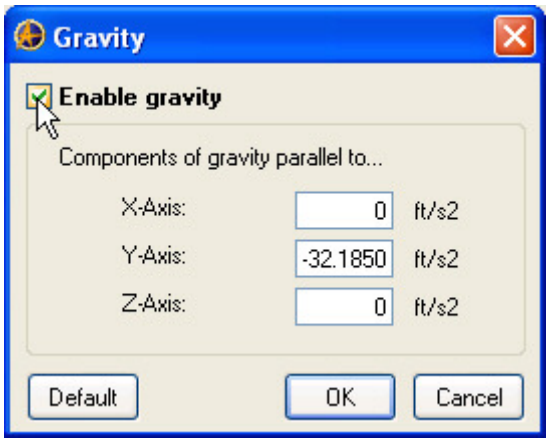
Note: The Gravity Node is shown grayed when gravity is not enabled.

1. Right-click on Gravity, and select **Enable Gravity**

Or

Right-click on Gravity, and select **Edit**. The Gravity dialog appears.

2. Check **Enable gravity** in the dialog.

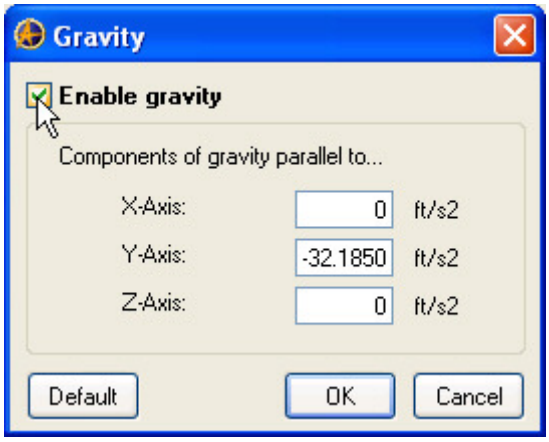


3. Click **OK**.

➢ ***To Change Gravity Settings***

1. Right-click on Gravity, and select **Edit**. The Gravity dialog appears.

2. Check **Enable gravity** in the dialog.



3. Specify the components of gravity along each major axis of the Assembly. Enter values in each of the X, Y, and Z – Axis text boxes.

4. If you want to go back to the default values for gravity, press the **Default** button. The default values are -9.81 m/s^2 or -32.185 ft/s^2 along the Y-Axis, and zero along the X- and Z-axes.

5. Click **OK**.

16.6 Simulation Types and Parameters:

To specify the Simulation Type and Parameters used for a Simulation, from the **Motion** main menu, select **Settings**. Choose the **Simulation Settings** Tab.

There are various methods used to produce simulations of systems and mechanisms, each with their own characteristics. Choosing the type of Simulator to use, and the parameters that specify it, can be very important to producing useful results:

- Dynamic Simulation
- Kinematic Simulation
- Quasi-Kinematic Simulation

The default settings provided by Alibre Motion will produce effective simulations in a very wide variety of situations. You should find them useful when simulating most mechanisms on everyday scales. If you are working with mechanisms that move or rotate very fast, or very slowly, are extremely large or small, you may get better results by adjusting these parameters:

- Time-Step (or Simulation Frame Interval)
- Error Tolerance

16.6.1 Simulation Types

Dynamic Simulation:

This is the most accurate and, therefore, realistic type of simulation. Each body in the system is analyzed to determine the physical characteristics that influence its motions and accelerations. The effects of any forces or torques applied to or within the system are calculated and applied precisely, and the movements and accelerations of each body are derived as accurately as possible within the specified simulation parameters. The correct solutions for a particular time-step are found using an iterative process, which can involve thousands and often millions of calculations for each simulation time-step or frame. By default, Alibre Motion uses this type of simulation.

Kinematic Simulation:

Kinematics is a branch of physics which is concerned with the motions of objects and particles without regard to the forces or torques which create those motions. The familiar laws of motion (such as $v = u + a \times t$) are examples of the application of kinematics. So a Kinematic Simulation is driven solely by any motions prescribed for the system, and does not take account of the weights or moments of inertia of the Parts involved. There are situations in which this may produce a useful simulation; for instance, if the reason for simulating is to detect the range of motion of a part, possible interferences, or for generating demonstration movies. Manually moving constrained Parts in Assemblies using the mouse can sometimes produce results similar to a Kinematic Simulation.

Though purely Kinematic Simulation may sometimes be acceptable, there are some cases in which it will produce a very different result from a Dynamic one. For instance, a piston and arm moving up and down will normally produce a smooth rotation in the crank to which they are connected. However, a purely Kinematic Simulation may show the crank rotating back and forth 180 degrees, never completing a full circle. Normally the angular momentum of the crank and arm would carry the motion through the “top-dead-center” position, but angular momenta are ignored in Kinematics. For this reason, Alibre Motion does not use this type of simulation.

Quasi-Kinematic Simulation:

The other type of simulation used by Alibre Motion is a new development of Kinematic Simulation. Rather than ignoring the mass, inertia and momentum of a Part, Alibre Motion substitutes values such as 1 wherever possible, which means that solutions to large and complex problems can often be found much more efficiently. This type of simulation may be best for very large or complex assemblies, when generating demonstration animations, detecting possible interferences, and where only the ranges of motion are the outputs of interest.

16.6.2 Producing Efficient and Useful Simulations

There are a few simple general principles that, when followed, will help produce accurate and efficient simulations. Some are good general CAD practice, while others are specific to simulating with Alibre Motion:

Build Full-Size

This is good general CAD practice, but is crucial to accurate simulation. Specifying a large force to act on a body may produce misleading results if the body is ten times shorter, and a thousand times lighter, than its real-world counterpart.

Specify The Material (or at Least the Density)

Alibre Motion needs to know the mass and the moments of inertia of a part in order to simulate it. These are calculated by Alibre Motion from the density of the Part, and the volume and shape of its design. Be sure to verify the correct material density has been specified for each part, whether you are designing new parts, or importing existing ones.

Group Non-Independent Parts into Sub-Assemblies

The efficiency of a simulator is always dependent on the number of moving parts it has to analyze and compute. In designs with Parts that are locked together and cannot move independently of one another, simulation will be more efficient if they are grouped into sub-assemblies that the simulator can treat as one object. No loss of simulation accuracy is to be expected in these cases, as the physical characteristics of a sub-assembly are calculated from its constituent parts. However in many cases, especially where there are numerous constraints and parts, the benefits in performance and efficiency can be significant.

Include Nuts and Bolts In Sub-Assemblies

An extension of the last point with regard to small items, whose dynamics are rarely of interest, is that it is usually best to include them in a subassembly with one of the parts they secure. For instance, you might want to “attach” bolts to one part, the nuts to the other, and then constrain those two sub-assemblies together as another sub-assembly. Alibre Motion will then only need to solve the dynamics of one part, rather than many.

Ensure The Constraints are Correct

Alibre Motion uses Automatic Constraint Mapping (ACM) to define the ways that Parts and Sub-Assemblies can interact. Ensuring that the Assembly moves as required before activating, Motion goes a long way towards ensuring the expected behavior from Simulations. Also, be careful not to over-constrain the Assembly, as this may cause locking.

Build Realistic Parts

Not just the mass, but also the shape and size of a part determines its moments of inertia, which influence the way it moves when acted on by a force or torque. For instance, a basketball has larger moments of inertia than an identical-weight soccer-ball has, and because of this, accelerates more slowly when rolling downhill. (Of course, if you drop them straight down, neglecting air-resistance, they would accelerate identically.) Similarly the center-of-mass must be in the right place to produce realistic behavior.

Use Realistic Constraints

Most constraints in Alibre Design represent a real-world physical behavior. For instance in the real-world, a rod passing through a sleeve is constrained by the contact between the outer surface of the rod and the inner surface of the sleeve. The Alibre equivalent of this relationship would be an Align constraint between the two surfaces. However, some useful constraints provided by Alibre Design do not have a real-world counterpart, such as those which include a Reference Geometry feature, an Axis or a Plane. Even though Alibre Motion may simulate the constraint such that the parts move exactly as expected, the Reference Geometry could lie outside the Part, and forces and torques may act through that point, which in reality could not happen. Avoid constraining against Reference Geometry in Assemblies you plan to simulate.

16.6.3 Automatic Constraint Mapping (ACM) in Alibre Motion

In the past, one of the most time-consuming processes when creating models for simulation was adding constraints. Often a CAD design had constraints that needed to be recreated from scratch in the simulation program. Alibre Motion automatically creates simulation constraints from the Design Constraints already present in the Assembly, using a process called Automatic Constraint Mapping, or ACM.

ACM is completed automatically when the Motion Explorer is activated from the Motion Menu, and whenever a new simulation is created or Regenerated.

Tip: Before activating Motion, ensure all constraints are correct and that the Parts and Sub-Assemblies move as required.

16.6.4 Moving and Fixed Parts

Automatic Body Mapping (ABM)

Alibre Motion uses Automatic Body Mapping to analyze your Assembly designs, and determine which Parts and Sub-Assemblies should become Moving Parts for the purposes of simulation. If a Sub-Assembly is **Made Flexible,** the Parts and Sub-Assemblies in it will all produce moving parts. Automatic Constraint Mapping will also create simulation constraints for each of its contained constraints. If the Sub- Assembly is not “Flexible,” the entire Sub-Assembly will become a Moving Part.

Because of this, it is common for the Motion Explorer to show that the simulation has more moving parts than the Design Explorer shows.

Fixed Parts

“Anchored” Parts and Sub-Assemblies will always become Fixed Parts. The dynamics of Fixed Parts are not calculated, and are used by Alibre Motion only when they are constrained to Moving Parts, as well as for determining any Interferences or Collisions.

Physical Characteristics of Parts And Sub-Assemblies

The physical characteristics of the moving parts in the simulation decide how they react to forces and torques applied to them. Alibre Motion calculates these characteristics directly from your existing Part and Sub-Assembly designs:

- Location
- Orientation

- Mass - calculated from the Material specified
- Location of the center of mass (also known as center of gravity, or CG)
- Moments of inertia
- The position of each point at which another body or force can act

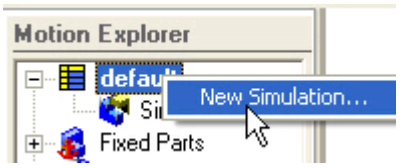
All of these are computed automatically by Alibre Motion from the Assembly, whenever a Simulation is generated. They come from the design of each part, and the constraints in the Assembly.

Tip: Ensuring that these characteristics are correct when designing your parts and assemblies is a crucial step in the process of producing accurate and useful simulations.

16.6.5 Creating Simulations

➢ ***To create a new simulation:***

1. In the Motion Explorer, right-click on the name of the assembly, and select New Simulation. The New Simulation dialog appears.



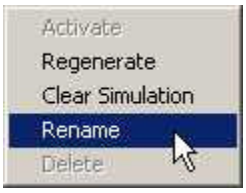
2. In Name, type a name for the new simulation.
3. In the Copy From drop down list, select the existing simulation you want to use as the starting point for the new one, or choose **None**.
4. Check the Active checkbox if you want to make the new simulation the active one.
5. Click OK to create the new simulation.

16.6.6 Renaming Simulations

Alibre Motion supplies a default name for each new Simulation you create. To help you keep track of them, it is often a good idea to give them more meaningful names, as you might with Design Configurations. For instance, you might name one “Motions Only," and another “Forces Active."

➢ ***To Rename a Simulation:***

1. In the Motion Explorer, right-click the Simulation you want to rename, and select **Rename**.



2. Type the new name for the Simulation.
3. Press **Enter** to save, or **Escape** to cancel the renaming. Simulations will be listed in alphabetical order in the Motion Explorer.

16.6.7 Running Simulations

After you have all of the necessary simulation options set, you can begin the simulation. To do this, make sure you have **Enable Simulation Mode** checked from the Motion main menu. Click the Play tool to begin the simulation. Choose the Pause tool , then Stop to end the simulation.

Optimizing Playback Performance

While the Alibre Motion Simulator is generating the simulation, playback will normally be slowed. These data are only calculated once for any given simulation, and playing the simulation back for the second time will be faster. A good visual indication is the Frame Slider Indicator on the Playback Deck: if the pointer is at the far right of the Slider, the Simulator is running, and performance should be expected to be slowed. If the pointer is running over an already blue section, the Simulator is not running.

- **Set the Target Playback Rate** - Make sure that the Target Playback rate is set to a reasonable value, using the Target playback frame rate check-box in the Playback Settings tab of the Settings dialog.

Note: This target rate will not be achievable in all cases.

- **Stop Other Programs** - If you have other calculation intensive programs running in the background, they may have some effect. Usually, programs that are not doing anything at the time will have a negligible effect, but often starting other programs will completely monopolize the CPU, slowing simulator performance.
- **Hide Reference Geometry** - Hiding Planes and Axes improves playback performance, as Alibre Design will not have to resize the features at each frame as the various parts of your assembly move. (Press **CTRL + SHIFT + P** on your keyboard to hide all the reference geometry in your model.)
- **Optimize Your Graphics Card performance** - The Graphics Card in your machine will have a great influence on the speed at which graphic images can be displayed. The manufacturer of the card or your computer may well have information on the best settings for your situation.
- **Suppress Irrelevant Parts** - Excluding parts which have no influence on the simulation may improve playback performance, especially if they are complex or irregularly curved shapes.
- **Close Information Windows** - Generating the XY-Plots of dynamic measurements, and generating Interferences demands a great deal from your computer, and you will usually find that hiding these, as well as the Motion Status Window, will give the best performance.
- **AVI Generation** - Generating AVI's will slow playback performance, and the amount to which they do so will vary greatly depending on the Video Handler or CODEC, that you use.

16.6.8 Simulation Warnings

While reading your Assembly design, and also before and during simulation, there may be times when Alibre Motion detects a condition which may affect the accuracy or some other aspect of the Simulation you want to perform.

For example, you will be warned when any Moving Parts in your assembly have extremely high or low densities, as this may be an indication that the results you get will not be realistic. In all of these cases, the Motion Status window opens automatically if it is not already showing, and the warnings are displayed. If you are satisfied that the situation described is as you intended, you can close the window, and ignore the messages, if you so choose.

Simulator Messages

Just before and during Simulation, the Alibre Motion Simulator displays various messages showing its status, and the progress of the simulation calculations. These are displayed in the bottom half of the Motion Status Window (unless you have turned them off by unchecking Show Simulation Messages on the General Tab of the Settings dialog). In some situations, such as when you specify a prescribed motion that cannot continue, the Simulator will stop and display various messages here. If the Simulator stops for no apparent reason, make sure that the Simulator messages are visible, as they often provide useful feedback about what has happened, and what may be done to correct the situation.

➢ ***To turn on and off the Motion Status window:***

From the **Motion** main menu, check the option **Show Status Window** to turn it on, and uncheck the option to turn it off.

16.6.9 Maintaining Multiple Simulations

Multiple Simulations

Alibre Motion allows you to maintain multiple Simulation Setups for *each* Configuration in your Assembly design. Each Simulation may have completely different Motors and Motions, as well as different Traces, Measurements and Simulation, and Playback settings specified. Using this feature, you can try out multiple scenarios on an assembly with, for example, different strength motors, and compare the results from each.

Simulations with Design Configurations

When Motion is enabled in the workspace of a new Assembly, a default simulation is created using Automatic Body Mapping and Automatic Constraint Mapping. The Part and Constraint data is taken from the Configuration currently active at that time, and the Simulation generated applies to that specific Configuration.

Note: Only Simulations for the Currently Active Configuration are displayed in the Motion Explorer.

Alibre Motion detects when you change the Active Configuration, and if it already has Simulation Setup data for that Configuration, the Motion Explorer will switch to that Simulation. If not, a new default Simulation Setup will be generated for the newly activated Configuration.

16.6.10 Deleting Simulations

If you have more than one simulation for a particular assembly, you can delete all of them except one. One simulation will always remain in the Motion Explorer.

➢ ***To delete a simulation:***

In the Motion Explorer, right-click the simulation you want to delete, and select Delete.

16.6.11 Activating a Different Simulation

If you have more than one Simulation Setup for the currently active Configuration, they are displayed under the Configuration name at the top of the Motion Explorer. Currently inactive Simulations are shown grayed out.

➢ ***To Switch Between Simulations:***

1. In the Motion Explorer, right-click the Simulation you to which you want to switch.

2. Select **Activate**. The Simulation is activated and checked, and any relevant Simulation Warnings are shown.

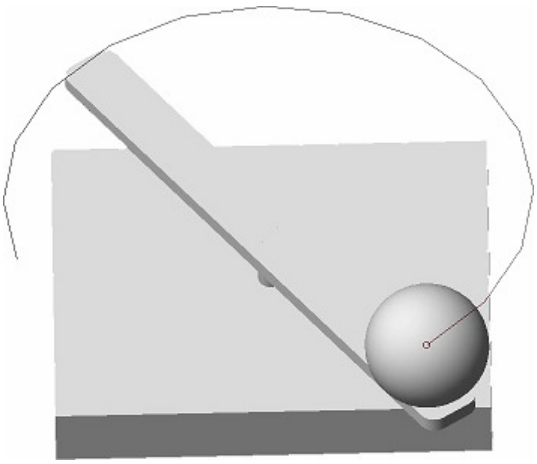
16.7 Results and Feedback from Alibre Motion Simulations

There are several different ways to obtain and view the results of simulations in Alibre Motion, each of which may be most useful in any given circumstance. Any combination of these may be used at the same time:

- Traces
- Video Generation
- X-Y Plots
- Dynamic Measurements
- Exporting Data

16.7.1 Traces - Visualizing Paths and Vectors

Traces can be extremely useful and efficient for visually examining simulation data. They enable you to see at a glance how a Part moves during the simulation, and to see how its velocity and acceleration changes. Traces may also add greatly to the visual impact of Videos generated from simulations of your designs, and Alibre Motion gives you several parameters that you can use to customize the way that Traces are displayed.



Traces are persistent 3D objects

Traces are shown as lines and shapes on the main canvas, but they are 3D reference geometry that dynamically change, enabling you to easily visualize 3D paths as you pan, zoom, and tilt during playback.

Trace Options are stored with the Simulation setup data in your Assembly file

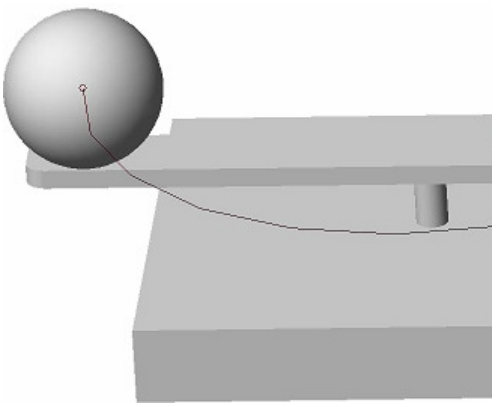
Unless you have chosen not to (by un-checking the “Save Simulation Setup Data in Assembly file” Settings option), then all information concerning your simulation, including your customized Trace options, are stored in the Assembly file. This means you can share not just an Assembly and Simulations, but also how they are displayed, for maximum effect.

Display Vector Data Using Traces

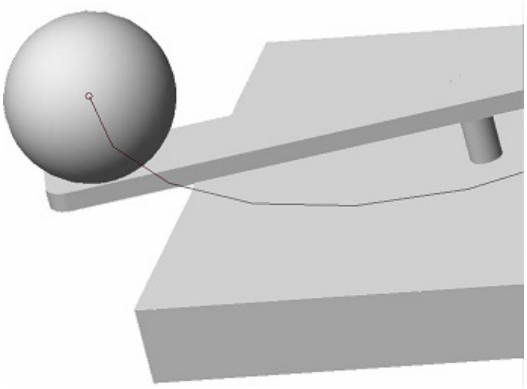
The Velocity and Acceleration of individual Parts can be displayed using traces. The direction of these vector quantities is shown as a line connected to the center of gravity (CG) of a part, or the Point of Action of a constraint. The length of the displayed line represents the magnitude of the Vector Quantity. Velocity vector lines are scaled logarithmically, such that Zero velocity is a line 5 pixels long, and the maximum possible velocity (C, the speed of light in a vacuum) is half the length of the diagonal extents of the assembly. To derive quantitative velocity and acceleration data, use X-Y Plots, or the “Current Value…” option.

Examples Of Various Trace Options

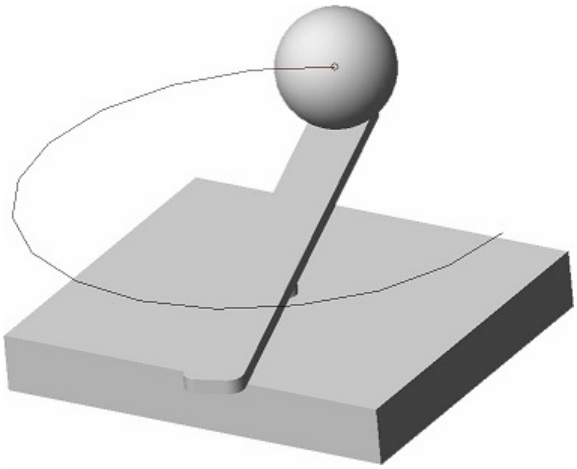
Here are some examples of the sorts of traces that can be produced using the options in the Trace Options dialog:



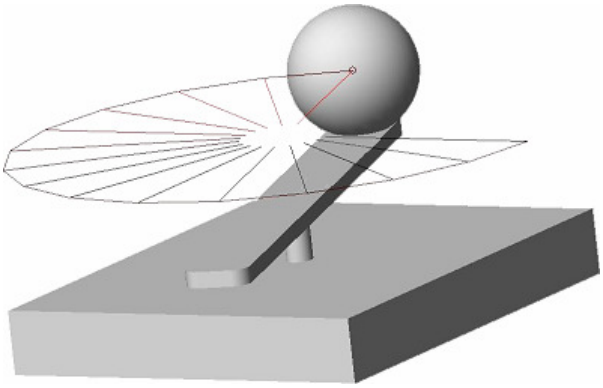
8 Frame Trail-length, Position, Velocity and Acceleration Traces Shown, Position Trails Shown



As above, with “Position - Fade Trails” selected



As above, with “Trail Length” of 30 frames



Acceleration trails shown

Creating Traces

➢ ***To create a Trace:***

1. In the Motion Explorer, right-click on a Moving Part or a Constraint, and select the type of Trace you want to display from the Trace menu. A new Trace item is shown below the Traces item in the Motion Explorer:

Note: Traces are shown on the main canvas during playback.

➢ ***To Change Trace Options:***

1. Right-click the trace you want to modify, and select or deselect the desired items. To change the way Traces of particular types are shown, select **Trace Options…** from the right-click menu of any Trace, or the **Traces** item in the Motion Explorer:

2. Change the desired options in the Trace Options dialog.

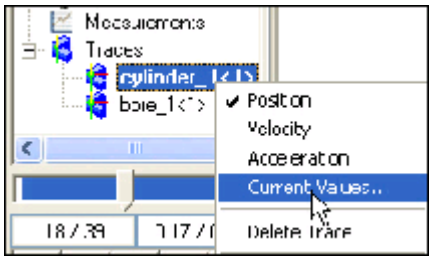
Current Values

➢ ***To see the current values of a Trace:***

1. You must have created a trace for a moving part or constraint, and have traces enabled.

2. In the Motion Explorer, expand the Traces category. You will see each of the moving parts or constraints that you for which you created a trace.

3. Right-click on one of the parts or constraints and select Current Values. The Data dialog appears.



This dialog shows the current values of any measurable quantity at the current frame. You cannot change any values here, and no selections you make reset the current simulation.

Notes:

Position data is shown relative to the Assembly origin.

The position, velocity and acceleration data shown for a Part is the data for its Center of Mass (CG)

- **Frame** - Shows the current playback frame at which the measurements are taken.
- **Data type** - select the type of data you are interested in. The selections in this drop down will vary depending on the item for which data is being shown. When you make a different selection, the relevant data is shown immediately.
- **X, Y, Z** - The data is shown as components parallel to the major axes of the Assembly design in the X, Y, Z boxes.
- **Magnitude** - The total magnitude is shown in the Magnitude box.

16.7.2 Generating Video with Alibre Motion

Generating Video (AVI) files can be a useful way of easily sharing the results of simulating your designs with colleagues and others, as generated files may be distributed via email, internet download and many other means, and may be played on a wide variety of systems.

Important Note: Not all CODECs installed on your system will be able to generate video. Though many are available as free downloads from Microsoft® and other manufacturers, some may only be capable of playing back video, and not generating it. To protect your system, always be extremely careful when downloading programs and other files from the internet, and only ever do so only from trustworthy sources.

How videos are created

Clicking the Video option button opens the Generate Video (AVI) Form, giving you the opportunity to specify the name of the file to be created, and various other settings that affect the video generation process. Then during playback, the main canvas is saved as a bitmap. Each bitmap is combined together and packaged into an AVI file. Whenever videos are generated on your system, the software that does this is called a CODEC (short for COder / DECoder), or Video Handler. These are generally small programs that need to be installed and registered, and it is likely that there are several of these already on your system. Alibre Motion is able to utilize those CODECs that handle a color-depth of 24-bits. The list of CODECs currently installed on your machine is shown in the drop-down box in the Advanced Tab of the Generate Video (AVI) Form.

The CODEC used to *play* a video must be compatible with the CODEC used to *generate* the video, so each AVI file holds information about how it was generated. A video player program examines the file, determines whether a suitable CODEC is already installed, and some video players may optionally download and install them automatically if one is not.

It is worth noting that the speed of generation, the quality of the generated video, and the number of systems on which it may be easily played are all heavily dependent on the choice of CODEC used to generate it. A great deal of information concerning CODECs and video generation is available online.

Note: During Video generation, some CODECs may show an options form or other display to allow you to monitor progress or change features. Playback may also be considerably slowed while videos are being generated. It is also possible that some CODECs may cause your system to slow or hang, though this is rare.

Not all CODECs can Generate Video

Since CODECs are often written by third parties, your operating system generally cannot distinguish between those that may be used to both generate and playback videos, and those that cannot, and are only useful for playback. Since it may also be hard to predict the effect of the Quality and Key- Frame parameters, it is worth experimenting to find settings that work best for you.

Generate Video Settings

General Tab

1. In **Output file to create**, specify the name and path of the video file to be generated. Click the **Browse** button ... to select a file and path from a standard File dialog.

2. In **Movie display name**, enter the name of the movie as it should be shown in a video playback program.

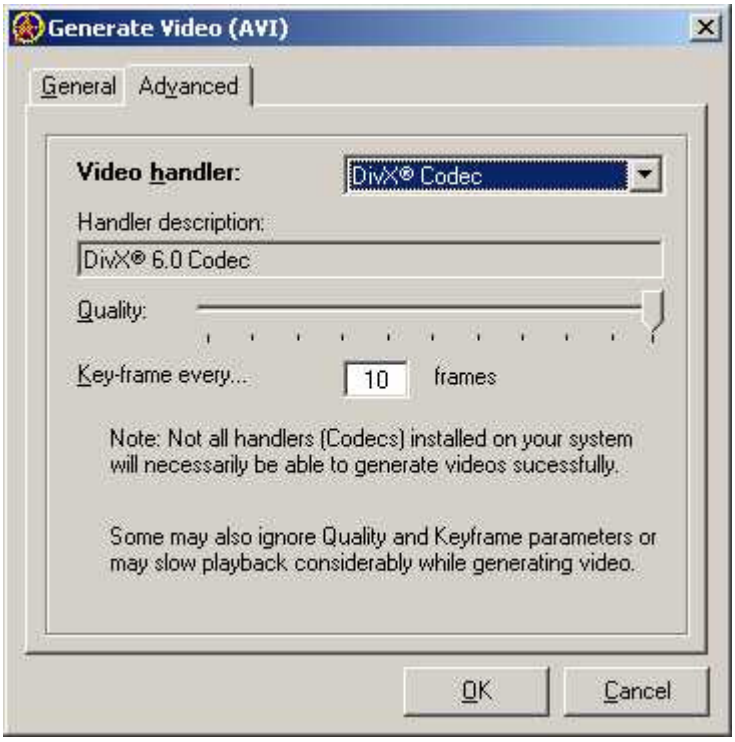
   Note: Not all video players display this name, or may do so only as an option.

3. In **Video frames per second**, enter the number of frames per second at which video playback programs should display the generated video.

   Note: The maximum speed at which video may be played back depends on the system and the video playback program.

4. Click **OK** to apply changes and close the dialog, or **Cancel** to abandon changes and close the dialog.

Advanced Tab



1. In **Video handler**, select the CODEC you want to use to generate the video (AVI) file. This drop-down list shows all the video handlers, or CODECs, that are installed and registered on your particular system. This list will vary from system to system, and some systems may not have any Codecs installed.

2. The **Handler description** field shows the internal description of the CODEC as provided by the CODEC itself.

3. The **Quality** parameter is used by some CODECs to determine the quality of the video to generate. Generally, the higher the quality, the larger the file size.

4. The **Key-frame every** field is used by some CODECs to determine how often the entire image should be stored within the generated video file. Some CODECs use a process whereby an initial frame (a key-frame) is stored, then each subsequent frame is stored only as the difference between that frame and the last. The purpose of this process is to minimize file-size.

5. Click **OK** to close the form. If a suitable CODEC, file, and parameters have been chosen, a video will be generated while the simulation is played. If you do not want to create a video, or save your changes, click **Cancel**.

16.7.3 Generating X-Y Plots from Simulation Data

You can generate X-Y Plots if you have selected Dynamic Measurements for at least one of your moving parts or constraints.

➢ ***To select a Dynamic Measurement:***

1. In the Motion Explorer, right-click on one of the moving parts or constraints, and select **Dynamic Measurement**. The Dynamic Measurement dialog appears.

2. In **Name**, enter a name for the measurement.

3. In **Measurement**, select which type of measurement you want to take for this part or constraint.

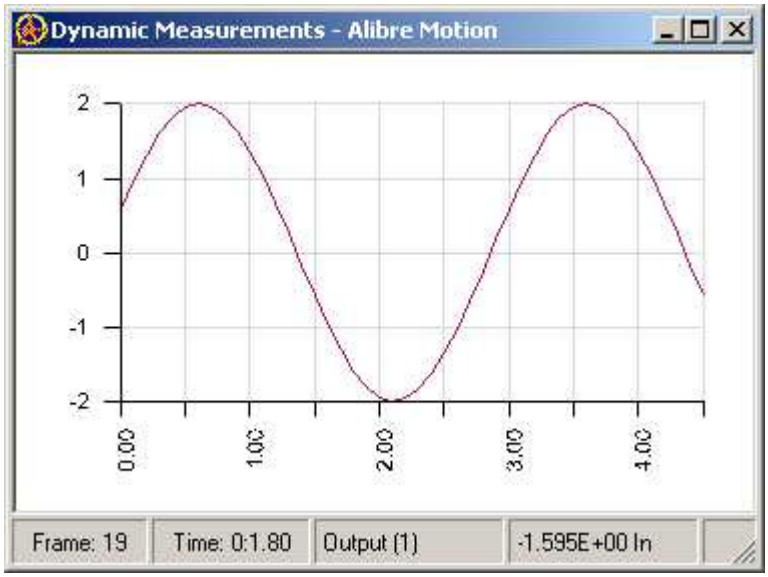
4. In **Component**, choose which component to measure - X, Y, Z, or total magnitude.

5. In **Display Color**, choose the color you want to represent this measurement when plotted.

6. Check the **Show in X-Y plot** option if you want this measurement to appear on the graph when you have X-Y plot turned on.

Once you have created your Dynamic Measurements, they will appear in the Motion Explorer under the Measurements category. Any measurements that you have selected to show in the X-Y plot will appear on the graph when you have the **Show/Hide X-Y Plots** button selected.

You can turn on the plot option before or after the simulation has been run.



➢ ***To Copy and Paste Chart Data:***

1. You can copy and paste the chart data from Alibre Design into another application for use in reports and presentations.

2. Right-click in the graph area and select **Copy**. You will see the following dialog:

Alibre Motion
Chart data has been copied both as text and as an image. Depending on the target application, you may need to use 'Paste Special' or similar to access the image.
Show me this again next time
OK

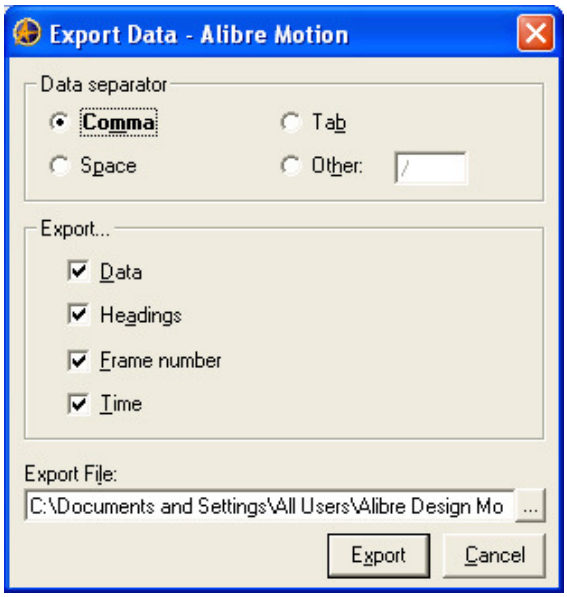
This tells you that, if you go into another application and choose Paste, you will be pasting only the values of the data in text. If you choose Paste Special, or a similar option, you will be given the choice to paste the text or an image. If you want to paste both text and the image, you will need to do it twice. Click **OK** to close the dialog.

3. Open the application where you want to paste the data and choose the appropriate **Paste** option.

➢ ***To Export Chart Data:***

You can export the chart data out of Alibre Design for use in reports and presentations.

1. Right-click in the graph area and select **Export chart data**. The Export Data dialog appears.



2. In **Data separator**, select which method you want to use to separate the data points.

3. In **Export**, select all of the options you want exported out.

4. In **Export File**, click the **Browse** button to browse to a location to save the file, and choose whether you want it saved as a text file or a .csv file.

5. Choose **Export** to export the data to the file you specified. The Export option exports only the chart data, not the plot image.

16.8 Detecting Interferences

While generating or playing back Simulations, Alibre Motion allows you to check for Interferences between parts at each frame. To enable this, select the Interferences tool on the Playback deck while in Simulation Mode. The Interferences dialog opens, which allows you to set options and view the details of any Interferences found.

Note: Interferences can be a calculation-intensive operation, since it checks for clashes with each part against every other part in your assembly. It is usually best to simulate first, and then check Interferences only during playback, because during simulation, the Simulator may be using a large part of the capacity of the CPU in your computer.

16.8.1 The Interferences Dialog

Note: Depending on the complexity of your model and the speed of your system, checking for Interferences can slow playback performance.

- Check Stop playback when found to automatically stop playback whenever any Interferences are found. This allows you to find the configuration where two parts first collide, for instance.
- Check Show interference extents to draw a red box around any interferences found at the current frame, enabling you to quickly identify which parts are interfering, and where.
- In Interferences found, Alibre Motion displays the number of interferences found in the assembly.
- The Interferences List displays the details of each interference. For each Interference, the names of the two parts are shown, as well as the volume of interference, which is the amount the two parts overlap.

16.9 Frequently Asked Questions (FAQ)

- What is Dynamic Simulation?

  Please see the *overview of simulation* (see "An Overview of Simulation" on page 313) for details on this question.

- What is the difference between Simulation and Animation?

  Simulation is the process of calculating the positions of the bodies at a given moment. Animation is displaying the bodies at the new positions in sequence.

- Does Dynamic Simulation involve stresses and strains within the bodies?

  No, generally Dynamic Simulation is concerned with what is termed Rigid-Body Dynamics, which assumes each body is infinitely strong, and infinitely stiff.

- How do I generate video of my Simulations?

  Please see the section on *generating videos* (see "Generating Video with Alibre Motion" on page 340) for information on how to do this.

CHAPTER 17

The Alibre Vault by M-Files

The Alibre Vault by M-Files is a complete data management solution. It is completely integrated into Windows and appears as a virtual drive, providing a familiar interface. The Alibre Vault has special integration with Alibre Design, allowing you to perform several tasks directly from within Alibre Design.

The Vault is composed of two components: the Vault Server and the Vault Client. For single users, both of these items will usually be installed on the same computer. While the Vault is powerful for a single user setup, its real power is realized when multiple users (each with their own Vault Client) are using a centralized Vault Server, allowing everyone to see and share the same data while never creating duplicates of items.

The Alibre Vault can be used with any and all of your programs, such as Excel, Word, PowerPoint, and any other program you may have. This allows you unparalleled freedom and control over your documents and projects.

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17.1 Description of Metadata

Metadata is what the Alibre Vault uses to categorize your data. In effect, it is a replacement for a folder structure, but its use goes far beyond simple file organization.

Metadata can basically constitute any property you can think of. Some basic properties include items such as File Type or File Name. Any items in the **File > Properties > General** tab of part or assembly workspaces are also available metadata for you to use. You can create new metadata types, such as "Export Restrictions" or "Launch Date", or anything else you might need to help you define CAD data, projects, work flows, press releases, contacts, or any other business-related item.

Metadata is used to organize your data, and is the most important Vault concept. All items within the Vault are differentiated based almost strictly on their metadata. Metadata is also used to search your Vault. The Alibre Vault comes with default metadata for each type of Alibre Design file. This metadata reflects items in the **File > Properties > General** tab of any workspace.

17.2 Differences Between the Vault and the File System

The Windows File System and the Alibre Vault are fundamentally different and operate completely differently. Internalizing the differences early on will save you a lot of time and make the transition to the Vault substantially easier.

The File System:

- Uses folders to store data
- Uses folders and file names to differentiate data
- Lets you browse a Folder Structure to find your data
- Allows only one file of a name to be in a particular folder
- Has no inherent versioning capability, check in/out, security, or other PDM capabilities.

The Alibre Vault:

- Does not use the concept of Folders to store data. Instead, all data is stored in the Vault Server, which is, in essence, a large database.
- Does not use the concept of Folders to differentiate data. Instead, an item's metadata is used to differentiate it from other files. File Name is a basic example of meta data.
- Does not let you browse a Folder Structure to find data. Instead, you browse something called Views. Views are essentially filters for metadata. For example, a view can show you things such as:
    - "All Alibre Part Files" (all vault items with a .ad_prt file extension)
    - "All projects due in the next two weeks" (all vault items with a Due Date < 2 weeks)
    - "All Products" (Shows all your different products and lets you drill down to show all associated part, assembly, drawing, BOM, Word, Excel, etc. files you have associated with a particular product).
- Allows infinite items of the same file name to be stored in the Vault. Remember, the Vault is a database and, much like you can have multiple identical rows in Excel, you can have multiple identical items in the Vault. However, since you will define metadata for each item, File Name is only one piece of a file's defining factors.
- Has inherent versioning, check in/out, security, notification, etc.

17.3 Enabling or Disabling the Alibre Vault

You should plan to exclusively use either the Vault or the File System. This ensures that your data is not duplicated. From the Home window, you can choose whether you want to save items to the Vault or the File System.

➢ ***To save all items to the Vault***

1. From the Alibre Design Home Window, select **Tools > Options**.

2. Select the **Use Vault** option and choose the Vault you want to use. It is recommended that you use the default Alibre Vault, since it contains a default metadata structure that mimics the Design Properties structure within Alibre Design (in the **File > Properties > General** tab of 2D and 3D workspaces). Also, it has a native concept of Alibre Design file types.

Note: The **Use Vault** option is only selectable if the Vault is detected as being installed and at least one Vault is available and selected.

➢ ***To save all items to the File System***

1. From the Alibre Design Home Window, select **Tools > Options**.

2. Make sure the **Use Vault** option is not selected.

Note: Items that currently exist on the File System will always be saved to the File System using the **Save** command. Only new items will be saved to the Alibre Vault if you have enabled the option. If you have existing files on the File System that you want to put onto the Alibre Vault, use the **Save As** or **Save As All** commands instead of **Save**.

17.4 Checking Items In and Out of the Alibre Vault

The Alibre Vault allows you to check items out for editing. This prevents concurrent editing of an item by multiple users. When you have checked out an item, other users can view it in Read Only mode.

Items are checked out:

- When you open an item within Alibre Design
- When you open an item within the Vault and select **Check Out**
- When you get into Edit mode on an item that is not checked out

  For example, when you open an assembly, only the assembly file is checked out. If you were go into Edit mode on one of the parts in the assembly, that part would be checked out at that time automatically.

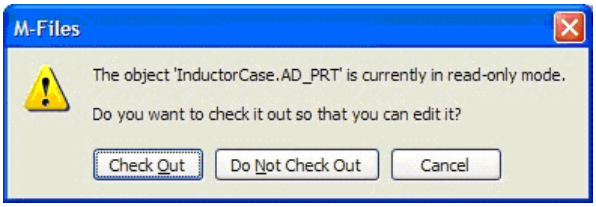
Items are checked in:

- When you explicitly check an item in inside the Vault interface
- When you check **Make Version** in the save dialog

  Make Version checks the item in, and then immediately checks the item back out so you can continue to edit it.

Opening an item that is not checked out

If an item is not checked out when you open it using the **File > Open** command within Alibre Design, the item is automatically checked out during the open function. If you want to open an item in Read Only mode, even though it is not checked out to someone else, browse to its location in the Vault outside of Alibre Design. When you try to open the file outside of Alibre Design, you will see the Check Out dialog below, which allows you to open the file in Read Only mode by selecting **Do Not Check Out**.


M-Files
The object 'InductorCase.AD_PRT' is currently in read-only mode.
Do you want to check it out so that you can edit it?
Check Out
Do Not Check Out
Cancel

Opening an item that is checked out

If the item is checked out to you, the file will open normally.

If the item is checked out to someone else, the file will open in Read Only mode.

Checking In Files

When you want to check in files, you must do so manually.

➢ ***To check in files***

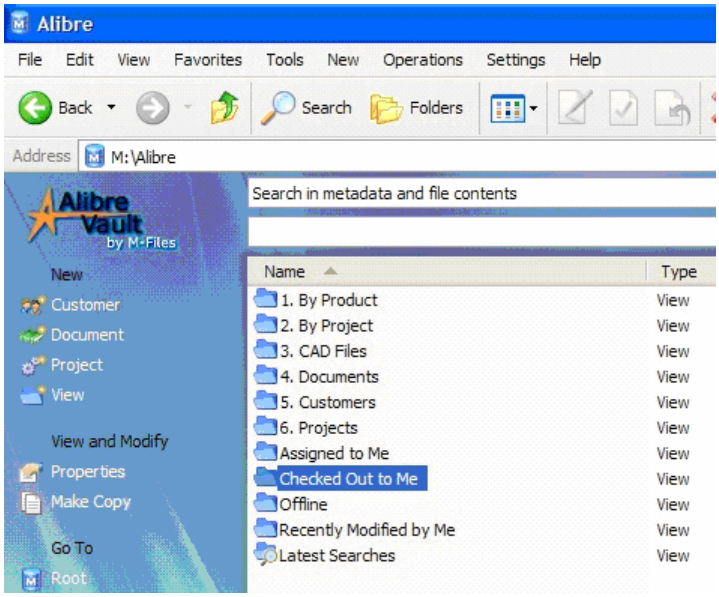
1. Browse the Vault interface through the Windows Explorer to find the files you want to check in.

2. Right click the files and select **Check In**, or press **Ctrl + I**.

➢ ***To find all files checked out to you***

1. Browse the root folder of the Vault.

2. Select the **Checked Out to Me** view. All files checked out to you are displayed. You can multi-select them, and then right click and select **Check In**, or press **Ctrl + I**.



17.5 Finding Items in the Alibre Vault

17.5.1 Searching

The Alibre Vault has several ways to sort and search your data. These various capabilities allow you to sift through extremely large amounts of data almost instantly.

Quick Search

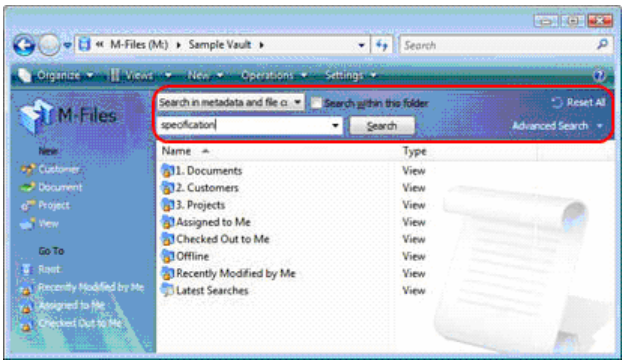


Figure 104: Quick Search

Quick Search is a useful way to search for documents and other objects in the Vault. Quick Search looks for objects that have the search word contained in the file contents or metadata. In the search field, enter a search string that has to do with the object, such as the title, the login name of a person who has edited it, or a customer related to it, and click the **Search** button. The search string does not need to be a whole word; you can truncate the word by using an asterisk (e.g., tender*). For example, if you have a part named Inductor.ad_prt you could find it by searching for ind*. Words and phrases searched for are saved in a pull-down menu, making it easy to repeat the searches.

With the menu items on the search bar, you can choose to search within document and object properties (metadata), file contents, or both. You can also restrict the search to the current folder. In the top right corner, there is a button with which you can clear all search criteria.

Quick Search Options

There are two search functions, simple and detailed. You can use advanced or detailed search criteria by selecting **Advanced Search** on the right-hand side. For more information, refer to *Detailed Search* (on page 352). The advanced or detailed search allows you to restrict the search. For example, you can search for all objects, for documents only, or for customers. You can search for documents from a specific project.

The following table lists the available Quick Search options:

All words	Objects found contain all specified search words.
Any word	The search will return all objects that contain at least one of the specified search words.
Boolean	The search allows you to use more specific search phrases and different operators.

You can use different expressions to find documents and objects that strictly meet your search criteria. The table below lists the search operators available for use in the Alibre Vault.

+ or -	Put a plus sign (+) in front of a search word to make sure that all of the documents and other objects found contain the word. Use a minus sign (-) to make sure that none of the documents and other objects contains the word. Example search: +technical &endash;construction
"phrase"	Enclose the search words in double quotation marks to find objects where the words appear adjacent to each other. Example search: "functional specification"
AND (boolean search only)	The AND operator combines two search terms. Documents found contain both terms. Example search: functional AND specification
OR (boolean search only)	The OR operator retrieves all documents containing at least one of the terms entered. Example search: agenda OR minutes

( ) (boolean search only) Brackets are used to group search terms together.

Example search: (agenda OR minutes) AND market*

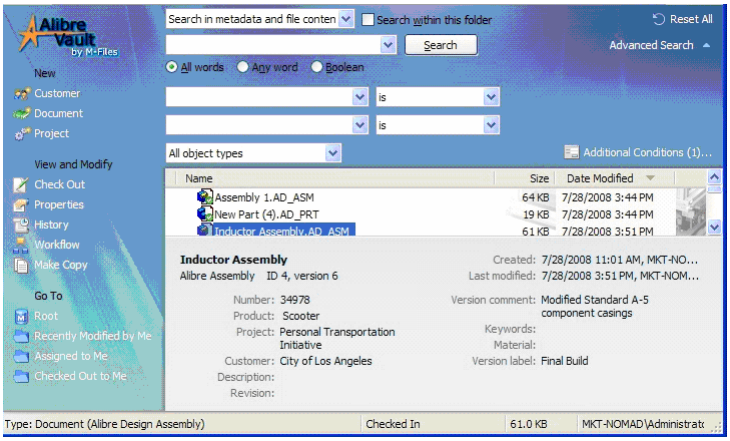
This search returns all objects that contain the words "agenda" or "minutes," and that also contain a word or words beginning with "market."

You can make the results more specific by selecting the **Search within this folder** option. When this option is selected, the folder is searched using the criteria from the previous search. This means that the search will also cover the objects found in the previous quick search.

The Alibre Vault ranks the search results by counting the number of occurrences of the search words in each document. The document with the greatest number of occurrences is displayed first.

The latest searches are saved in a specific view. This allows you to easily return to previous search results later on in the day.

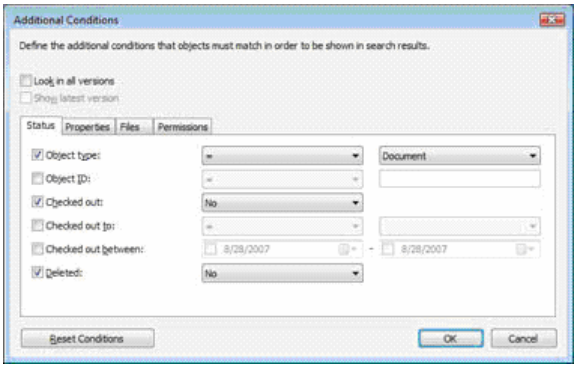
Detailed Search



With the detailed / advanced search, you can define more specific search criteria. The more search criteria you use in the advanced search, the more likely you are to find the exact object you want. This allows you to prevent the search from returning too many results.

For example, you can select a project definition from the property list. Each object has property values that are assigned to it via the document card. These property values can be used to search for documents in a precise manner. The document property could be Project, and the value of this project could be "Hospital Expansion (Florida)". If you perform an advanced search with these values, the search returns all documents for which "Hospital Expansion (Florida)" has been specified as the value of the Project property. You can also restrict the search to a specific object type, such as customers or contact persons.

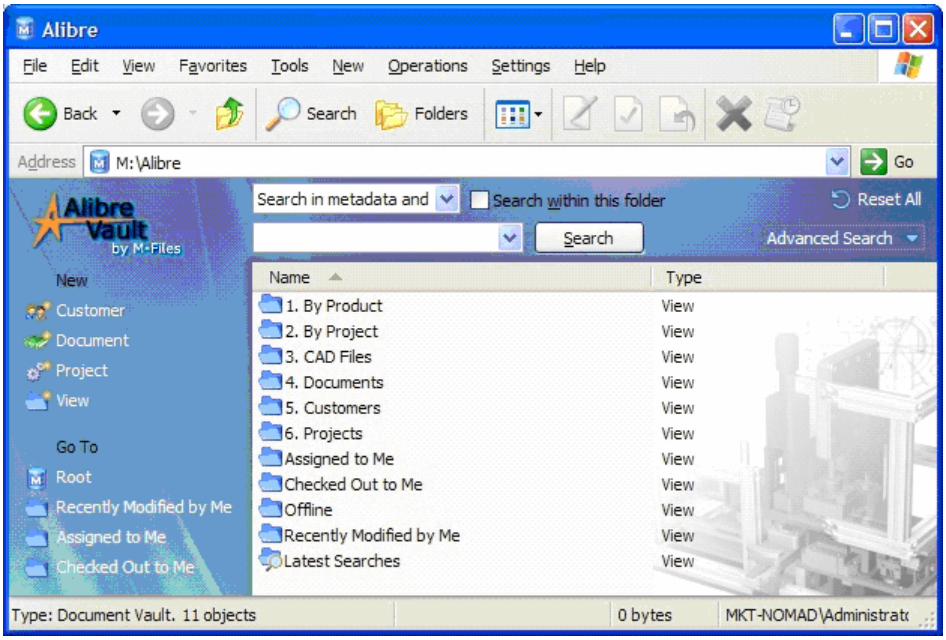
If necessary, you can also define additional detailed search conditions.



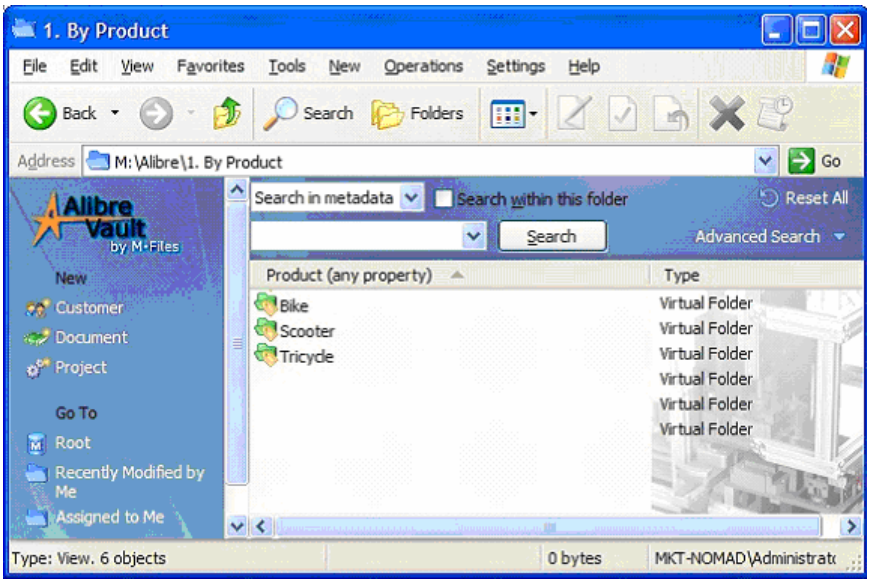
17.5.2 Views

Views are one of the most important concepts in the Alibre Vault. They are logical views of your entire data set, essentially providing the functionality of filters based on the metadata you have entered for all of the items in your Vault.

Several default views are set up in the Alibre Vault, as shown below.



When you click on a view, the Alibre Vault searches all your data based on the criteria that defines the view. For example, a By Product view will search all your data and find every Product value that has been used. It will then display all available projects for you to drill deeper.



Clicking on a Product listed in this view will show all documents, such as CAD files, Word documents, Excel sheets, PDFs, etc., that have been associated with the selected Product.

17.6 Renaming Files

You can rename files in the Vault without implication on your designs. Alibre Design relies on data beyond the file name to determine which parts, assemblies, drawings, and BOMs go together.

Notes: You must use the Vault interface, not Alibre Design, to rename files.

Alibre Design does not rely on metadata to link files. This means that you can change any metadata without affecting constituent links of your design (for example, which parts belong to an assembly or drawing).

➢ ***To rename a file:***

1. Find the file in the Vault.
2. Make sure the file is checked out to you.
3. Right click the file and select **Rename**.
4. Type the new name for the file and press **Enter**.

17.7 Copying Items

You can copy items in the Vault using one of two methods:

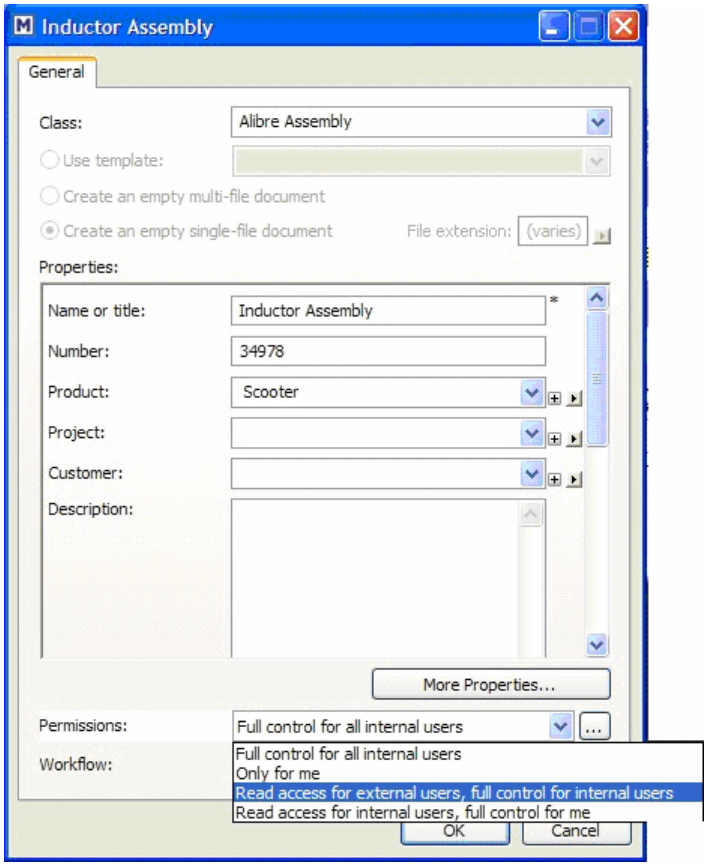
1. Open the item in Alibre Design, and perform a **Save As** operation.
2. Find the file in the Vault interface, right click it, and select **Make Copy...**

17.8 Using Permissions

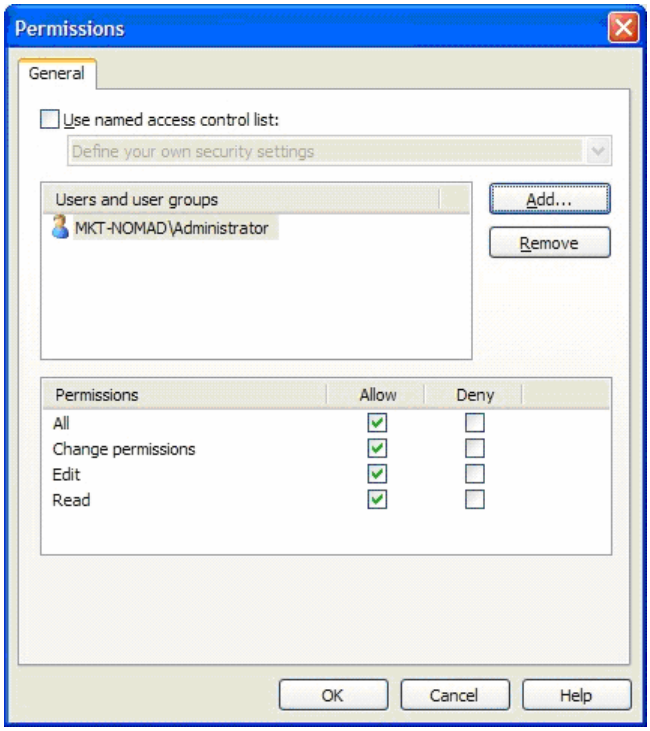
Setting the permissions of a document in the Vault is done directly on the Meta Card for any item. The Vault Server provides the available users or user groups available for granting or denying permissions. Instructions for setting up new users and user groups on the server are available in the Vault Help system, accessed by right clicking the M icon on the lower left of your screen in the System Tray.

➢ ***To set permissions while saving***

1. Open the item's meta card
2. If you are saving the document, you will see a **Permissions** drop down that allows you to set preset permissions. If you need to add additional permissions, click **...** next to the Permissions line.



3. After clicking the **...** item next to the **Permissions** drop down, you will see the full permissions dialog.



4. Uncheck **Use names access control list** if you want to define a specific user permission that is not available from the drop down. When you do this, **Add...** becomes available. You can use **Add...** to select specific users or user groups to grant permissions.

5. Set the specific permissions you want for each user or user group by changing the **Allow** or **Deny** check marks for each permission item.

6. Click **OK**.

➢ ***To set permissions in the Vault interface***

Setting permissions is identical in the Vault interface. Select a file, right click it, and select **Properties**. Instead of a **Permissions** drop down, there will be a **Permissions** Ttb at the top of the meta card that provides the same functionality described above.

17.9 Setting Up Notifications

You can set up notifications for several events within the Vault. For example, you can set up groups or individuals to be notified when actions such as Check In, Check Out, Version Rollback, etc. are performed.

To enable notifications, you will need to access the Server Administrator and input your mail server's SMTP settings, along with any authentication that is required by your server. Refer to the Vault help system, accessed by right clicking the M icon on the lower left of your screen in the System Tray.

17.10 Versioning

17.10.1 Versioning Concepts

Using the Vault, you can create versions of documents. Versions serve several purposes, such as providing a history of the design development and providing the ability to revert to a previous version.

When versions are created

Versions are created in two scenarios:

1. You manually check something into the Vault

2. You check **Make Version** in the save dialog when saving something to the Vault. When you do this, the item marked for a version is checked in and then checked back out automatically so you can continue editing the item.

Note: Versions are not created every time you save an item. They are only created when an item is checked into the Vault, or when you explicitly select **Make Version** from the Save dialog.

Where to find the current version information

While browsing the Vault, you can see the version information of any item by clicking on it. The properties of the item will be shown near the bottom of the window.

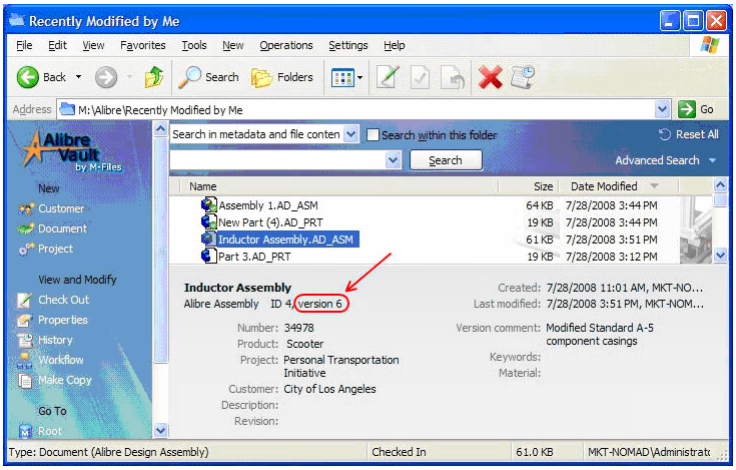


Figure 105: Item Properties Showing Versions

17.10.2 Viewing Version History

➢ ***To view all the versions for an item***

1. Find the item in the Vault for which you want to view the version history.

2. Right click it and select **History...** The History dialog appears, displaying all versions of the document.

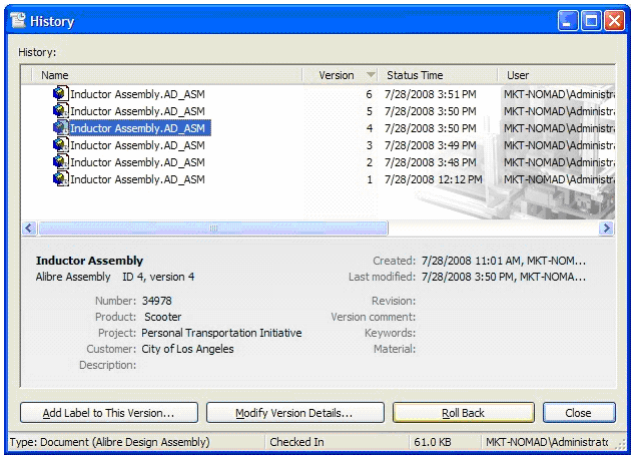


Figure 106: Version History for an Item

17.10.3 Creating Version Labels

With the Vault, you can create Version Labels for your versions. Version Labels are useful for marking milestones in an item's creation. For example, you can have standard version labels such as "Meets Export Restrictions," "Initial Concept," or "Final Build." The label will be displayed under **Version Label** in the Version History window, as well as in the item's **Property Display** area directly in the Vault interface.

➢ ***To assign a label:***

1. Open the Vault interface and find the item for which you want to create a label.

2. Right click the item and select **History...** or press **Ctrl+H**. All of the items versions are shown.

3. Select the version of the item for which you want to create a label.

4. Click **Add Label to This Version...**

5. Select the **Version Label** you want from the drop down. You can also add a new version label to the list by clicking the small arrow next to the field and selecting **Add New Value to List**. By default, no labels are assigned to any version.

6. If you want the selected label to be cleared from the other versions too, check the **Clear this version label from the other versions of the object** box. This is useful when, for example, you want only one version of a contract to be labeled as approved.

   The version label is visible from within the History dialog, or on the item's property display area near the bottom of the Vault interface.

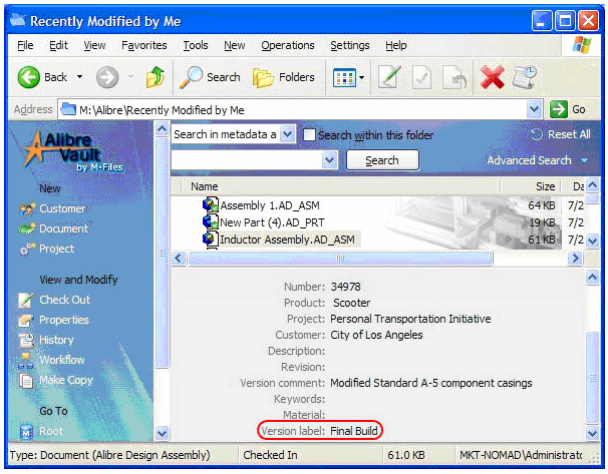


Figure 107: Version Label for an Item

17.10.4 Rolling Back to Previous Versions

As a default, the different versions of an object are listed from the oldest to the newest on the basis of the creation date. You can restore old versions of documents with the **Roll Back** function. This function creates a new version of the document with the contents of the old version. This is useful when you have made and saved changes in a document but do not want them to take effect.

Note: You can use the **Roll Back** function only when the document is not checked out.

➢ ***To roll back to a specific version***

1. Find the item in the Vault you want to roll back.

2. Right click the item and select **History...** The History dialog appears, displaying all versions of the document.

3. Select the version to which you want to roll back, and then click the **Roll back** button.

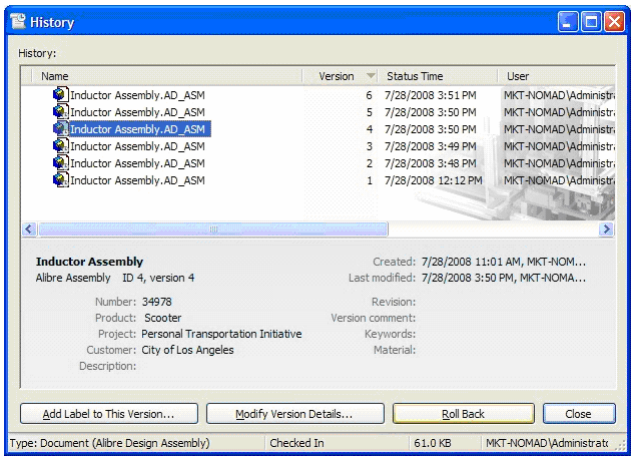


Figure 108: Roll Back to Previous Version

4. A confirmation dialog appears. Click **Yes**.

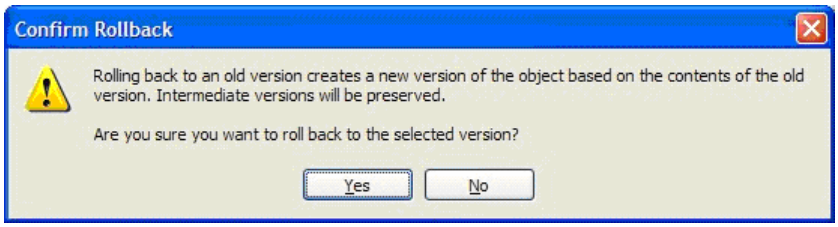

Confirm Rollback
Rolling back to an old version creates a new version of the object based on the contents of the old version. Intermediate versions will be preserved.
Are you sure you want to roll back to the selected version?
Yes
No

Figure 109: Roll Back Confirmation Message

A new version is created, containing the contents of the old version. All intermediate versions are maintained.

17.11 Relationships

In the Alibre Vault, you can see the relationships between items (i.e., what items use other items). For example, you can find what assemblies use a specific part. These relationships between parts, assemblies, drawings, and BOMs can be seen using the Relationships dialog within the Alibre Vault.

Relationships are populated during an Alibre Save operation only, and only when an item is checked in. This means you will need to select the **Make Version** option during a save at least once for any given design in order for the Relationships to populate.

The two tabs that you will use are:

- **Relationships from this Object** - Shows what the item in question is using. For example, shows which parts are used in an assembly.
- **Relationships to this Object** - Shows what is using the item in question. For example, shows which assemblies use a part.

➢ ***To view an item's relationships***

1. Open the Alibre Vault interface by browsing to your Vault.

2. Find the item for which you want to see Relationships.

3. Right click the item and select **Relationships...** The Relationships dialog appears.

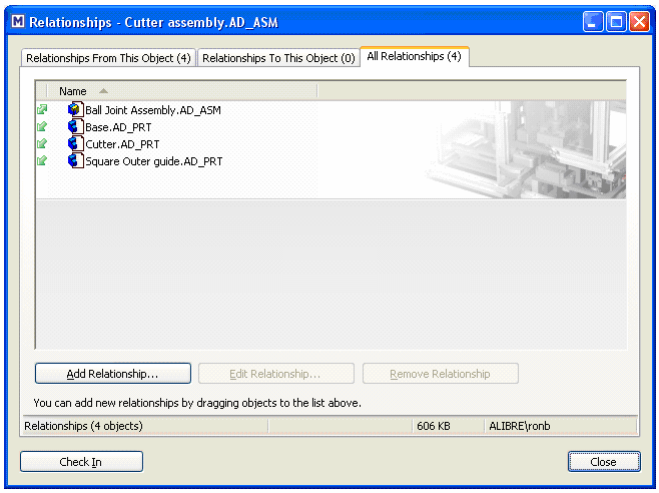


Figure 110: Relationships Dialog

This dialog contains the following tabs:

- **Relationships From this Object** - represents items that make up the item currently selected. In the example below, we are looking at an assembly file. All the Relationships in this case are in the **Relationships From this Object** tab, which means we're looking at the parts and subassemblies that comprise Cutter Assembly.ad_asm, and that nothing else, such as a drawing, is using the assembly.

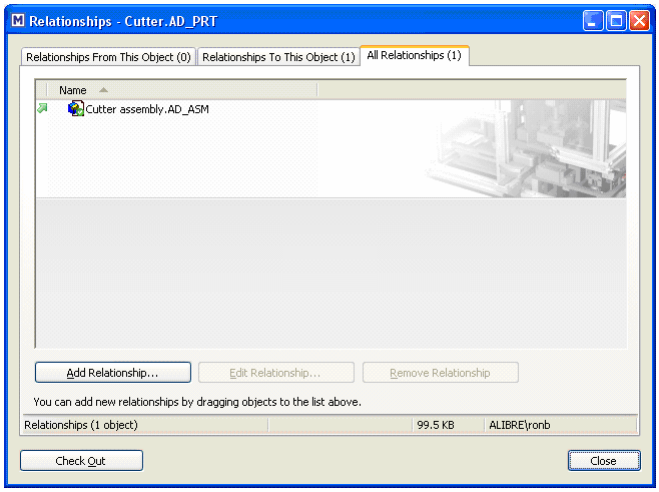


Figure 111: Relationships Example

- **Relationships to this Object** - represents where the item we are looking at is being used. In the example below, we're looking at cutter.ad_prt, one of the parts that is used in the assembly example above. We can see that it has one relationship, and that it points to Cutter assembly.ad_asm.

Limitations of Relationships

Relationships can only see "one level deep" in either direction. For example, if you have several parts that make up an assembly, and then create a drawing of that assembly, the parts will only know they are being used in the assembly, but will not have a relationship to the drawing. Likewise, the drawing will know it is using the assembly, but will not know about relationships to the specific parts within the assembly.

You can double click on items that show up in the results list to drill deeper and overcome this limitation. For example, if you need to see all the drawings that use a part, you would:

1. Open the Relationships dialog for the part

2. Select the **Relationships to this Object** tab to find out what items are using the part.

3. Click on any assemblies that use the part to open their Relationship screen.

4. View any drawings that use the assembly, and, therefore, the original part in question.

17.12 Getting Help

The Alibre Vault is a very deep product with many available options. We have attempted to cover the most common workflow issues here. However, there may be several additional use cases you want to learn about.

There is a complete Help system for the Alibre Vault, accessed by right clicking the icon on the lower left of your screen in the System Tray.

The Alibre Vault Help is comprehensive, and includes advanced topics such as:

- Setting up advanced user scenarios
- Setting up regular backups of the Alibre Vault Server
- Changing or adding new Meta Data to the data structure
- Connecting the server to your existing database

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